

Utah State University

DigitalCommons@USU

All Graduate Theses and Dissertations

Graduate Studies

12-2010

Plasticity and Damage in Bimodal Grain Size Al-5083: Microstructural Finite Element

Steven Nelson

Utah State University

Follow this and additional works at: <https://digitalcommons.usu.edu/etd>



Part of the [Mechanical Engineering Commons](#)

Recommended Citation

Nelson, Steven, "Plasticity and Damage in Bimodal Grain Size Al-5083: Microstructural Finite Element" (2010). *All Graduate Theses and Dissertations*. 717.

<https://digitalcommons.usu.edu/etd/717>

This Thesis is brought to you for free and open access by the Graduate Studies at DigitalCommons@USU. It has been accepted for inclusion in All Graduate Theses and Dissertations by an authorized administrator of DigitalCommons@USU. For more information, please contact digitalcommons@usu.edu.



PLASTICITY AND DAMAGE IN BIMODAL GRAIN SIZE AL-5083: MICROSTRUCTURAL
FINITE ELEMENT

by

Steven Nelson

A thesis submitted in partial fulfillment
of the requirement for the degree

of

MASTER OF SCIENCE

in

Mechanical Engineering

Approved:

Dr. Leila Ladani
Major Professor

Dr. Stephen Whitmore
Committee Member

Dr. Thomas Fronk
Committee Member

Dr. Byron R. Burnham
Dean of Graduate Studies

UTAH STATE UNIVERSITY
Logan, Utah

2010

Copyright © Steven Nelson 2010

All Rights Reserved

ABSTRACT

Plasticity and Damage in Bimodal Grain Size Al-5083:

Microstructural Finite Element

by

Steven Nelson, Master of Science

Utah State University, 2010

Major Professor: Dr. Leila Ladani

Department: Mechanical and Aerospace Engineering

Bimodal and nanocrystalline aluminum alloys are being investigated as stronger replacements for conventional polycrystalline aluminum alloys. Higher strengths are achieved by reducing the grain size of a metal; however, as the grain size is reduced the ductility diminishes. One solution that limits this decrease in ductility is the addition of a few microcrystalline grains into a nanocrystalline alloy, creating a bimodal microstructure that offers a better balance of strength and ductility. Two- and three-dimensional microstructural finite element (FE) simulations of monotonic and fatigue failures in Al-5083 having bimodal grain structures are conducted. To reduce the computational time and facilitate the modeling of microstructural features, a global-local model is developed. Macroscopic linear elastic and nonlinear plastic properties for each of the bimodal compositions are first used to simulate the tensile and fatigue tests in a global FE model. Subsequently, a local model that represents a single element at the center of the global model is built with distinct coarse grains (CGs) distributed throughout an ultra-fine grain (UFG) matrix. Ten percent of the elements in this model are defined as CGs, after which nanocrystalline and polycrystalline properties are assigned to the UFG and CG regions, respectively.

Available fatigue test data is utilized to generate a low cycle fatigue damage model for bimodal grains size Al-5083 and obtain the damage model constants for varied levels of coarse

grains. This fatigue damage model is then used in conjunction with a finite element continuum damage modeling approach, namely, successive initiation, to predict the damage and crack initiation sites and propagation paths in bimodal grains size alloys. The successive initiation method is used to continually accumulate damage in elements and initiate and propagate the crack through grains that reach the failure criteria defined for monotonic and cyclic loading. It is observed from the monotonic FE model that cracks initiate on the boundaries between CGs and UFGs then propagate through the UFG matrix around the CG until they become large enough to extend into the CGs themselves. In the cyclic FE models, the crack is observed to initiate in a CG and propagate along the CG and the surrounding UFG matrix.

(162 pages)

DEDICATION

First and foremost, to my advisor and teacher, Dr. Leila Ladani, for the hours of brainstorming and research direction. You kept me on track when things got too busy and got me back on track whenever an avenue of research was of no immediate use. I would not have finished without the numerous comments, corrections and questions you provided and posed.

To Jafar, James, and Eli, whose lengthy conversations kept me sane and cleared my head after long days of fruitless work.

To Dr. Fronk and Dr. Whitmore, who always made their classes enjoyable and enthusiastically supported this thesis every step of the way.

To Bonnie, who is still tirelessly answering every question I have with a smile on her face. I could have figured it out on my own, but you made it so much clearer and stress free. Thanks.

To my parents, who are always there for anything I need. For the long conversations when things are their hardest and for your interest in my highly technical world.

To my fiancé, Becca, who spends her days caring for our daughter while I sit in front of a computer. I promise all the engineering frustrations that I force you to listen to will pay off one day. Thanks for listening even when the topic is way over your head, and I love you.

And last, but most definitely not least, Lizzy, my 20-month-old bundle of joy. All the frustrations of a long day immediately fade in comparison to your exclamation of “Daddy!” and the hug and kiss I get every night when I walk through the door. Daddy is the proudest title I will ever have.

ACKNOWLEDGMENTS

This research has been supported by the National Science Foundation Civil, Mechanical and Manufacturing Innovation Division, Award No. 0927319. Additionally, microstructural images were obtained at the Boise State Center for Materials Characterization, whose work is supported by the National Science Foundation Major Research Instrumentation Program, Award No. 0521315.

Steven Nelson

CONTENTS

	Page
ABSTRACT	iii
DEDICATION.....	v
ACKNOWLEDGMENTS	vi
LIST OF TABLES.....	x
LIST OF FIGURES	xi
NOMENCLATURE	xv
INTRODUCTION	1
LITERATURE REVIEW	4
Processing and Fabrication.....	4
Cryo-milling	5
Degassing	6
Consolidation	7
Plastic Deformation.....	8
Microstructural Characteristics	9
Material Properties	11
Tensile Properties.....	11
Fatigue Properties.....	14
Failure Mechanisms.....	16
MATERIAL MODELS	18
Tensile Plasticity Model for CIPped Al-5083	18
Tensile Plasticity Model for HIPped Al-5083	20
Fatigue Damage Model	21
FINITE ELEMENT MODEL.....	27
Global-Local Modeling	27
Modeling Limitations	31
Successive Initiation.....	32
Successive Initiation in Monotonic Loading.....	33

Successive Initiation in Cyclic Loading.....	33
FINITE ELEMENT RESULTS.....	35
Tensile Failure of CIPped Al-5083	35
Two-Dimensional Model	35
Three-Dimensional Model	38
Tensile Failure of HIPped Al-5083	41
Two-Dimensional Model	42
Three-Dimensional Model	44
Cyclic Failure of HIPped Al-5083	47
Two-Dimensional Model	47
Three-Dimensional Model	50
Effect of Poisson Ratio Change During Plastic Deformation	52
SUMMARY AND CONCLUSIONS	55
REFERENCES	58
APPENDICES	63
Appendix A: Low resolution Finite element results.....	64
Tensile Failure of CIPped Al-5083	64
Two-Dimensional Model	64
Three-Dimensional Model	68
Tensile Failure of HIPped Al-5083	70
Two-Dimensional Model	71
Three-Dimensional Model	73
Cyclic Failure of HIPped Al-5083	75
Two-Dimensional Model	75
Three-Dimensional Model	78
Appendix B: ANSYS Input Files	82
Monotonic 2D Input Files	82
CIPped Global Model.....	82
CIPped Global Solution	84
CIPped Local Model	85

CIPped Local Solution	89
HIPped Global Model	91
HIPped Global Solution	93
HIPped Local Model	94
HIPped Local Solution	98
Monotonic 3D Input Files	99
CIPped Global Model.....	99
CIPped Global Solution	101
CIPped Local Model	102
CIPped Local Solution	107
HIPped Global Model	109
HIPped Global Solution	111
HIPped Local Model	112
HIPped Local Solution	116
Cyclic 2D Input Files.....	117
Global Model	117
Global Solution	120
Local Model	121
Local Solution	125
Cyclic 3D Input Files.....	129
Global Model	129
Global Solution	132
Local Model	133
Local Solution	137
Poisson Ratio Sensitivity Solutions.....	141
Change in CG Properties.....	141
Change in UFG Properties	143
Change in Bimodal Properties.....	145

LIST OF TABLES

Table		Page
1	Percent Element Composition of Al-5083	1
2	Reported Microstructural Characteristics of UFG and Bimodal Al-5083	11
3	Reported Tensile Properties of UFG and Bimodal Al-5083	12
4	CIPped Al-5083 Voce Constants	19
5	CIPped Voce Material Parameters for a Range of CG Contents [19].....	19
6	Tensile Properties of UFG, CG, and Bimodal Al-5083 Used in Fatigue Tests [14].....	20
7	HIPped Al-5083 Voce Constants	20
8	HIPped Voce Material Parameters for a Range of CG Contents	20
9	Fatigue Material Constants for UFG, CG, and Bimodal Al-5083	25
10	Coffin-Manson Power Law Parameters Used to Predict Fatigue Life.....	26

LIST OF FIGURES

Figure		Page
1	Optical Micrographs showing the interconnected porosity of CIPped billets degassed at (a) 415°C, (b) 440°C, and (c) 500°C [5].....	7
2	TEM images of CIPped and extruded Al-5083 with 10% CGs: (a) low magnitude view of CG band amid UFG matrix, (b) medium magnitude view of UFG matrix, (c) high magnitude view of CGs, and (d) high magnitude view of UFG matrix.	10
3	Stress strain curves showing (a) serrated hardening effects as a function of tensile strain rate for 10% CG sample [6], and (b) strain softening behavior of samples high CG contents[18].....	13
4	Fracture surface of 10% CG Al-5083 showing (a) shear fracture plane and cavitations, and (b) dimpled fracture surface.....	17
5	Stress strain curves for bimodal, UFG, and CG Al-5083 developed using Joshi's Voce hardening model.....	19
6	Tensile stress-strain curves for UFG, CG, and bimodal Al-5083 used for fatigue modeling.	21
7	Cyclic stress strain curves for UFG, CG, and bimodal Al-5083.....	25
8	Coffin-Manson plot of fatigue life for UFG, CG, and bimodal Al-5083.....	26
9	Global model of dogbone tensile specimen with loads and constraints described.	28
10	Local 2D model with randomly sized CGs in random locations.	29
11	Local 3D model with two large CG bands: one at the symmetry intersection of the model and one at the center of the model.	30
12	Crack initiation in tension corresponding to a gage section stress of 560 MPa (load step 1) in the global model; (a) eliminated elements in the microstructure, (b) stress fields in the UFG matrix and CG regions, (c) and (d) enlarged views of the areas indicated in (a).	36
13	Crack propagation in tension corresponding to a gage section stress of (a) 583 MPa (load step 3), (b) 595 MPa (load step 4), (c) 618 MPa (load step 6), and (d) 688 MPa (load step 12) in the global model.....	38
14	Crack initiation in tension corresponding to a gage section stress of 595 MPa (load step 9) in the global model; (a) eliminated elements in the	

	microstructure, (b) stress fields in a UFG matrix cross section taken at the x-location of the initial crack.	39
15	Crack propagation in tension corresponding to gage section stresses of (a) 612 MPa (load step 13), (b) 616 MPa (load step 14), (c) 625 MPa (load step 16), and (d) 690 MPa (load step 30) in the global model.	40
16	Crack initiation in tension corresponding to a gage section stress of 387 – 394 MPa (load steps 2 – 3) in the global model; (a) eliminated elements in the microstructure, (b) stress fields in the UFG matrix and CG regions, both after the third load step, and (c) and (d) enlarged views of the areas indicated in (a) after the third and second load steps, respectively.	42
17	Crack propagation in tension corresponding to a gage section stress of (a) 401 MPa (load step 4), (b) 408 MPa (load step 5), (c) 415 MPa (load step 6), and (d) 422 MPa (load step 7) in the global model.	43
18	Crack initiation in tension corresponding to a gage section stress of 380 MPa (load step 1) in the global model; (a) eliminated elements in the microstructure, (b) stress fields in the UFG matrix relative to CG locations.	45
19	Crack propagation in tension corresponding to gage section stresses of (a) 387 MPa (load step 2), (b) 394 MPa (load step 3), and (c – d) 401 MPa (load step 4): (c) interiors of CGs and (d) full model after failure.	46
20	Crack initiation and propagation in cyclic loading after (a) 100 simulated cycles, and (b) 125 simulated cycles. Enlarged views, (c) and (d), of areas indicated in (a).	48
21	Crack propagation in cyclic loading after (a) 150 simulated cycles, (b) 175 simulated cycles, (c) 200 simulated cycles, and (d) 325 simulated cycles.	49
22	Crack initiation in cyclic loading after (a) 500 simulated cycles, and (b) 550 simulated cycles.	50
23	Crack propagation in cyclic loading after (a) 700 simulated cycles, (b) 900 simulated cycles, (c) 1050 simulated cycles, and (d) 1250 simulated cycles.	51
24	Crack initiation in tension corresponding to a gage section stress of 584 MPa (load step 3) in the global model; (a) eliminated elements in the microstructure, (b) stress fields in the UFG matrix and CG regions.	65
25	Crack propagation in tension corresponding to a gage section stress of 595 MPa (load step 4) in the global model; (a) eliminated elements in the microstructure, (b) stress fields in the UFG matrix and CG regions.	66
26	Crack propagation in tension corresponding to gage section stresses of (a) 607 MPa (load step 5) and (b) 619 MPa (load step 7) in the global model.	66

27	Crack propagation in tension corresponding to gage section stresses of (a) 666 MPa (load step 10) and (b) 690 MPa (load step 12) in the global model.	67
28	Crack initiation in tension corresponding to a gage section stress of 599 MPa (load step 10) in the global model; (a) eliminated elements in the microstructure, (b) stress fields in a UFG matrix cross section taken at the x-location of the initial crack.	68
29	Crack propagation in tension corresponding to gage section stresses of (a) 612 MPa (load step 13) and (b) 616 MPa (load step 14) in the global model.	69
30	Crack propagation in tension corresponding to gage section stresses of (a) 620 MPa (load step 15) and (b) 690 MPa (load step 30) in the global model.	70
31	Crack propagation in tension corresponding to a gage section stress of 401 MPa (load step 4) in the global model; (a) eliminated elements in the microstructure, (b) stress fields in the UFG matrix and CG regions.	71
32	Crack propagation in tension corresponding to gage section stresses of (a) 408 MPa (load step 5) and (b) 415 MPa (load step 6) in the global model.	72
33	Crack propagation in tension corresponding to gage section stresses of (a) 422 MPa (load step 7) and (b) 429 MPa (load step 8) in the global model.	72
34	Crack initiation in tension corresponding to a gage section stress of 380 MPa (load step 1) in the global model; (a) eliminated elements in the microstructure, (b) stress fields in the UFG matrix on the edges of the CGs.	73
35	Crack propagation in tension corresponding to gage section stresses of (a) 387 MPa (load step 2) and (b) 394 MPa (load step 3) in the global model.	74
36	Crack propagation in tension corresponding to gage section stresses of 401 MPa (load step 4): (a) crack relative to CGs and (b) failed regions within the CGs.	75
37	Crack initiation in cyclic loading after 200 simulated cycles; (a) eliminated elements in the microstructure, (b) sites of maximum accumulated damage.	76
38	Crack propagation in cyclic loading after 400 simulated cycles; (a) eliminated elements in the microstructure, (b) sites of maximum accumulated damage.	77
39	Crack propagation in cyclic loading after (a) 1000 simulated cycles and (b) 6000 simulated cycles.	77
40	Crack initiation in cyclic loading after 400 simulated cycles; (a) eliminated elements in the microstructure, (b) sites of maximum accumulated damage.	78

41	Crack propagation in cyclic loading after 1200 simulated cycles; (a) eliminated elements in the microstructure, (b) sites of maximum accumulated damage.....	79
42	Crack propagation in cyclic loading after (a) 1600 simulated cycles and (b) 2600 simulated cycles.....	80
43	Crack propagation in cyclic loading after (a) 2800 simulated cycles and (b) 3000 simulated cycles.....	80
44	Crack propagation in cyclic loading after (a) 3400 simulated cycles and (b) 3800 simulated cycles.....	81

NOMENCLATURE

σ_Y	Yield strength
σ_0	Characteristic stress used in Hall-Petch relationship
k	Material constant used in Hall-Petch relationship
d	Grain diameter
σ_{pl}	Plastic (inelastic) stress
R_0, σ_S	Material constants used in Voce hardening law
ε_{pl}	Plastic (inelastic) strain
R_∞	Material constant used in Voce hardening law
b, ε_c	Material constants used in Voce hardening law
$c_1 - c_6$	Material constants used to determine Voce material constants
f_{cg}	Coarse grain fraction contained in bimodal material
$\Delta\sigma/2$	Stress amplitude
$\Delta\varepsilon_{pl}/2$	Plastic strain amplitude
K'	Material constant used in cyclic stress strain relationship
n'	Material constant used in cyclic stress strain relationship
F	Grain boundary strengthening factor used in fatigue model
C	Grain boundary constraint factor used in fatigue model
$\Delta\sigma_{eff}/2$	Effective stress amplitude
$\Delta K_{eff}/2$	Effective stress intensity factor amplitude
r_{CPZ}	Size of the cyclic plastic zone
r_{FDZ}	Size of the fatigue damaged zone
λ	Cyclic plastic zone correction factor
π	Mathematical constant relating any circle's circumference to its diameter
a	Instantaneous crack length
σ	Instantaneous stress
r	Distance away from crack tip
ε_{pl}^*	Accumulated plastic strain
σ^*	Critical stress value
V_{int}	Interaction strain energy

V	Volume of cyclic plastic zone
σ_{UTS}	Ultimate tensile strength
ΔJ	Integral representation of crack growth
N, N_{cyc}	Number of cycles undergone
N_f	Number of cycles to failure
$\Delta CTOD$	Crack tip opening displacement
a_i	Initial length of crack
a_f	Final length of crack
D^*	Damage accumulated after the current finite element solution
D_0	Damage accumulated in previous finite element solutions

INTRODUCTION

Aluminum alloys are known for being lightweight and ductile, though in general, not incredibly strong. This study focuses on Al-5083 which is a balance of Aluminum (Al) alloyed primarily with Magnesium (Mg). Besides already being one of the stronger Al alloys, Al-5083 is suitable for use at low, even cryogenic temperatures. It is resistant to corrosion at temperatures less than 60°C, and as a result is very commonly used in marine and amphibious applications along with many aerospace functions. Table 1 shows the acceptable ranges of metallic alloying elements that Al-5083 is comprised of.

Table 1: Percent Element Composition of Al-5083

Al	Si	Fe	Cu	Mn	Mg	Cr	Zn	Ti
92.55 - 94.25	0.4	0.4	0.1	0.4 - 1.0	4.0 - 4.9	0.05 - 0.25	0.25	0.15

Recent advancements in ball milling techniques, namely cryo-milling, produce very fine grains in metallic microstructures when employed. Materials produced in this manner are termed nanocrystalline or ultra-fine grain (UFG) materials, and demonstrate a marked increase in strength. This increase in strength is described very well by the Hall-Petch relation described by Eq. 1 where the yield strength, σ_Y , is inversely proportional to the square root of the grain size, d , where σ_0 and k are material constants [1,2].

$$\sigma_Y = \sigma_0 + \frac{k}{\sqrt{d}} \quad \text{Eq. 1}$$

Though the increase in strength is very much desirable, it comes at a price. Not only are the fabrication processes required very expensive, a feasible way to implement them on a large scale is yet to be developed. But even more costly than the price-tag, the reduction in grain size

severely limits dislocation motion, and therefore plastic deformation and ductility. In general, UFG materials fail very soon after yielding in a notably brittle manner. A multitude of ways to find an optimal level of strength and ductility are currently being investigated; most of them use different methods of grain coarsening to produce a material not quite as strong but much more ductile. They range from heat treatments [3,4], and varying the temperatures used in the fabrication processes [5,6], to precisely controlled in situ versions of the fabrication processes developed to significantly reduce the number of voids, inclusions, and other imperfections [7]. The first two of these essentially coarsen all the grains throughout the UFG material and achieve slightly better levels of ductility while maintaining most of the strength. The third is actually successful at maintaining the UFG size and the strength of the UFG material while increasing its ductility by a factor of four relative to values of tensile elongation at failure reported most frequently. But, the in situ technique proved to be even more costly and even harder to implement on a large scale.

Modifying the UFG material's microstructure to create a bimodal metal is the most promising means by which to achieve the optimal balance of strength and ductility. Bimodal microstructures are most frequently created deliberately – sometimes varying the fabrication process will unintentionally produce a bimodal microstructure [8] – by mixing un-milled alloy powders with the cryo-milled powders. The resulting microstructures have distinct micrometer sized coarse grain (CG) regions dispersed throughout an UFG matrix where the grain sizes are in the tens to hundreds of nanometers. As the volume fraction of un-milled powders is increased, the material's strength decreases as its ductility increases. Basically, the UFG region retains its strength while higher levels of deformation are allowed to occur in the weaker CG regions. Bimodal metals created via mixing un-milled and cryo-milled powders have properties that are reproducible as long as all other fabrication process variables remain unchanged.

Given the importance of finding stronger versions of the lightweight materials used in marine and aerospace applications, it is no surprise that the UFG version of Al-5083 is being heavily investigated. Though the findings presented here are centered around UFG and bimodal Al-5083, many of the results, particularly the increased strength and reduced ductility of UFG alloys and the better balance achieved by making them bimodal, apply to all UFG and bimodal alloys.

This study investigates the crack nucleation site and propagation path within the microstructure of bimodal Al-5083 using successive initiation techniques within a finite element analysis. Since bimodal metals have two distinct microstructural characteristics, one strong and brittle, the other weak and ductile, the location of failure relative to the two regions is not immediately apparent. One may assume that the nucleation site will be on the boundary of the CGs and the UFG matrix, since the property mismatch will produce large stress concentrations there. But, which region will fail first, and in what direction will the crack propagate from there? Also, since the applied loads can vary as much as the microstructure, the crack nucleation site and propagation path are investigated for both tensile and cyclic loading scenarios. This may significantly change the location of initial failure and the failure path. The results show that bimodal Al-5083 has a marked difference in failure mechanisms given the two loading scenarios. Since this is the case, some conclusions may be made as to whether making a UFG material bimodal is useful in all loading applications. Also, conclusions about optimum levels of CG content given a particular loading scenario are considered.

LITERATURE REVIEW

The ultra-fine grain and bimodal metals field of research is still very young. The advent of cryo-milling and the subsequent consolidation and forming techniques have only been around for about a decade. Since that time, researchers have been investigating the microstructural and material properties attained using these novel fabrication processes. The effect of different variations in the fabrication process and their effect on the microstructure and constitutive properties of UFG Aluminum, Copper, and Nickel are largely documented, and are summarized here.

Processing and Fabrication

The same variability in microstructural characteristics and material properties occur in both UFG and bimodal alloys, and so the overall process is described with only minor distinctions described between the two. As mentioned previously, the primary step in creating UFG materials is the cryo-milling of the alloy powders down to the nano-scale. After cryo-milling and any blending of un-milled and cryo-milled powders, the powders are degassed. Degassing removes as much of the remaining cryo-milling agents (Nitrogen and Stearic Acid) as possible, leaving as pure alloy powders as can be achieved [5].

Once cryo-milled and degassed, the powders must be compressed, or consolidated, back into a more conventional form of metal. Of course this can be done with purely cryo-milled powders or a blend of un-milled and cryo-milled powders when bimodal alloys are desired. Cold-isostatic pressing (CIP) and hot-isostatic pressing (HIP) are the most widely used consolidation techniques, though quasi-isostatic forging (Ceracon forging) is sometimes used. Consolidation serves to compact all the powders into an 80 – 100 percent dense billet, but many prior particle boundaries remain after isostatic pressing. If these are not removed, the as consolidated material will be severely limited in its ductility [9].

The final step of processing serves to remove these prior particle boundaries by means of an applied shear stress. Any form of plastic deformation, such as extrusion, rolling, or forging, will impart the necessary shear stress, break up the prior particle boundaries, and additionally, eliminate any lingering porosity that exists after degassing and consolidation [10]. Each step of the fabrication process is outline further in the following sections.

Cryo-milling

In conventional ball-milling, also known as mechanical alloying, particles of a given metallic alloy are broken up into microscopic powders via severe plastic deformation. The alloy powders and the milling media, typically stainless steel balls, are loaded into the milling apparatus which rotates about its horizontal axis, causing both the balls and powder to tumble over onto each other. Standard ball to powder ratios lie at about 32:1. Any powder that ends up between two balls as they collide is deformed or broken into finer powders. It is possible for multiple particles of powder to be pressed back together during a collision. Also, there is always the chance that the powder will weld itself to a ball it collides with. These counterproductive and undesirable occurrences are reduced when milled at cryogenic temperatures.

To cryo-mill powders essentially all that is needed is the addition of liquid nitrogen. The balls, powder, and nitrogen are all mixed together to form a liquid nitrogen ‘slurry,’ to which fresh nitrogen is continuously added for the entire milling time (approximately 8 hours). Stearic acid is also be added to the mixture as a process control agent to mediate the welding of particles [11]. When the powder and milling media are cooled to cryogenic temperatures several advantages over room temperature milling are observed. As alluded to previously, the recombination of powders and the welding of powders to the milling media is suppressed. Also, the liquid nitrogen environment significantly reduces the level of oxidation that can occur. And the milling time required to obtain nano-scale particles is largely reduced [9,10]. Immediately

following the completion of cryo-milling, the powder-nitrogen slurry must be transferred to an inert environment while the nitrogen evaporates away and then sealed to protect the powder from contamination.

Degassing

For the most part, little attention is paid to the degassing portion of the process when HIP is used for consolidation. But it is extremely important in achieving fully dense UFG samples when CIP is used for the consolidation step. It is the step in which the excess nitrogen particles, stearic acid, and other gaseous contaminants are extracted from the cryo-milled powders. The powders are transferred into a steel can, then ramped up to and held at an elevated temperature (300 – 450 °C) for a prescribed amount of time (2 – 24 hours) while under vacuum pressure conditions.

Intuitively, one can anticipate grain growth in materials wherever high temperatures or pressures are attained and held for any extended period of time. In UFG and bimodal materials, this grain growth is highly undesirable, though necessary to facilitate proper consolidation and densification. One study investigates the grain growth and density of CIPped and forged billets of Al-5083 when the degassing temperatures and hold times are varied [5].

Degassing temperatures of 415°C, 440°C, and 500°C are reached and held for 5.5, 10, and 2 hours, respectively, on three different samples of the same material. The average grain size after fabrication increased by 6.2%, and 14.7%, when the two materials degassed at higher temperatures are compared to the material degassed at 415°C. Additionally, the density increased from 98.4% (low temperature), to 98.8% (medium temperature) and 99.1% (high temperature). But the most telling observation was that the interconnected porosity of the as CIPped billet decrease significantly as the degassing temperature was increased, shown in Figure 1. The interconnected pores allow unwanted gaseous products to seep into and get trapped in the microstructure during the deformation step, which can affect the properties considerably.

Ultimately, when using CIP, higher degassing temperatures are highly desirable as its contribution to grain growth is small when compared to its ability to help eliminate the interconnected porosity and increase the final density of the consolidated billet [5].

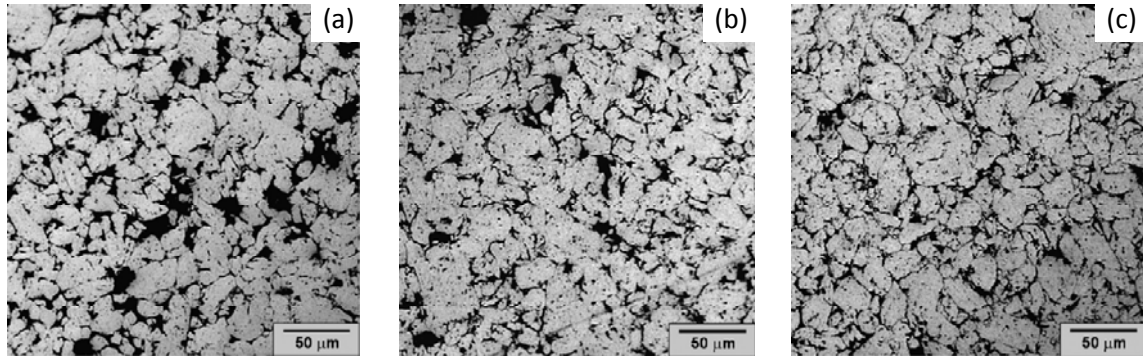


Figure 1: Optical Micrographs showing the interconnected porosity of CIPped billets degassed at (a) 415°C, (b) 440°C, and (c) 500°C [5].

This step receives less attention in literature describing HIPped materials because high temperatures are attained during hot-isostatic pressing, which serves to complement the high temperatures of degassing and produce a fully dense consolidated billet with very limited interconnected porosity. Of course, given the same degassing procedure, the UFG sizes of a HIPped sample can be 75% larger than a CIPped sample [3,12,13].

Consolidation

The process of compacting cryo-milled or blended powders back into a billet of material is termed consolidation in the vernacular of powder metallurgy and UFG/bimodal metals. Hot- and cold-isostatic pressing are the primary techniques by which UFG materials are consolidated, though there is also quasi-isostatic (previously referred to as Ceracon) forging [8,14] and spark plasma sintering [15]. The latter two techniques do not require a final deformation step like CIP

and HIP do, though sometimes CIPped or HIPped materials are subsequently quasi-isostatic forged in lieu of a secondary deformation step (extrusion, rolling, forging) [13,16,17].

To consolidate the cryo-milled powders they are loosely packed into a can. As the pressure, and temperature if HIP is used, is increased, the interconnect regions of porosity that inevitably exist between the powders are reduced as contact points between individual powders grow into necks. Eventually there are very few interconnected regions, and the pores themselves shrink as the consolidation continues.

Cold-isostatic pressing is the less expensive of the consolidation methods. It is conducted at room temperature and very high pressures of 300 MPa or greater [5,16-18]. Alternatively, hot-isostatic pressing is conducted at temperatures ranging from 275°C to 350°C and pressures ranging from 100 MPa to 200 MPa [6,19-23]. Some research has been done at temperatures and pressures outside the normal pressure ranges like 7 MPa [12] and 400°C [8], but the resulting microstructures and material properties do not vary from the normal range.

Plastic Deformation

Finally, the consolidated billets must be plastically deformed to break up the prior particle boundaries and eliminate any porosity formed during consolidation. These boundaries serve to catastrophically reduce ductility in UFG materials that already suffer from a lack of ductility and a shear stress must be applied to break them up. Several methods of shear stress application are frequently used, though extrusion is by far the most common. Rolling, forging and equal channel angular press (ECAP) are all also viable means of plastic deformation. All the aforementioned plastic deformation techniques can be performed at room temperature, or after first heating the material.

This step also serves to create a transversely isotropic bimodal material, especially when extrusion or rolling is used. Ultra-fine grains are small enough that any elongation in the

deformation direction is negligible, but when the material has a bimodal microstructure consisting of UFGs and CGs, the CGs tend to elongate significantly in the direction of deformation. The difference in properties in the longitudinal (parallel to the direction of deformation) and transverse (perpendicular to the direction of deformation) directions is not extreme, but definitely notable [18,24].

Forged materials do not exhibit such a high level of anisotropy [8,13], though there is still some. The directions of anisotropy are not so simple to define since forged materials are deformed in random directions, and some portions of the material tend to be deformed more than others. In some cases, particularly when the material undergoes quasi-isostatic forging, there is virtually no anisotropy, and the material can be treated in an isotropic manner [16,17].

In summary, there are four steps, each requiring high levels of precision, in the fabrication of UFG and bimodal alloys via cryo-milling. Microstructural characteristics and mechanical properties can vary widely with cryo-milling time, degassing temperature, and choice of consolidation process and subsequent plastic deformation technique. Care must be taken to choose the proper combination of steps to obtain the microstructural and properties desired.

Microstructural Characteristics

The end result of all the precise processing described previously is a microstructure comprised of very small grains that provide a substantial increase in strength as described by the Hall-Petch relation shown previously in Eq. 1. Typical UFG and CG diameters average around 200 nanometers and 2.0 micrometers, respectively, and have respective standard deviations of around ± 100 nanometers and ± 1.0 micrometers. There are notable amounts of deviations with fabrication process variations, as well as slight differences between multiple materials processed using identical fabrications steps. Representative microstructural images are shown in Figure 2. These images are of a bimodal sample of CIPped and extruded Al-5083 containing 10% CGs and

were taken at the Boise State Center for Materials Characterization on their JEOL 2100 transmission electron microscope (TEM). Grain sizes depicted here fall within the standard bimodal range and the extrusion direction is visible in Figure 2(a) by noting the elongated band of CGs.

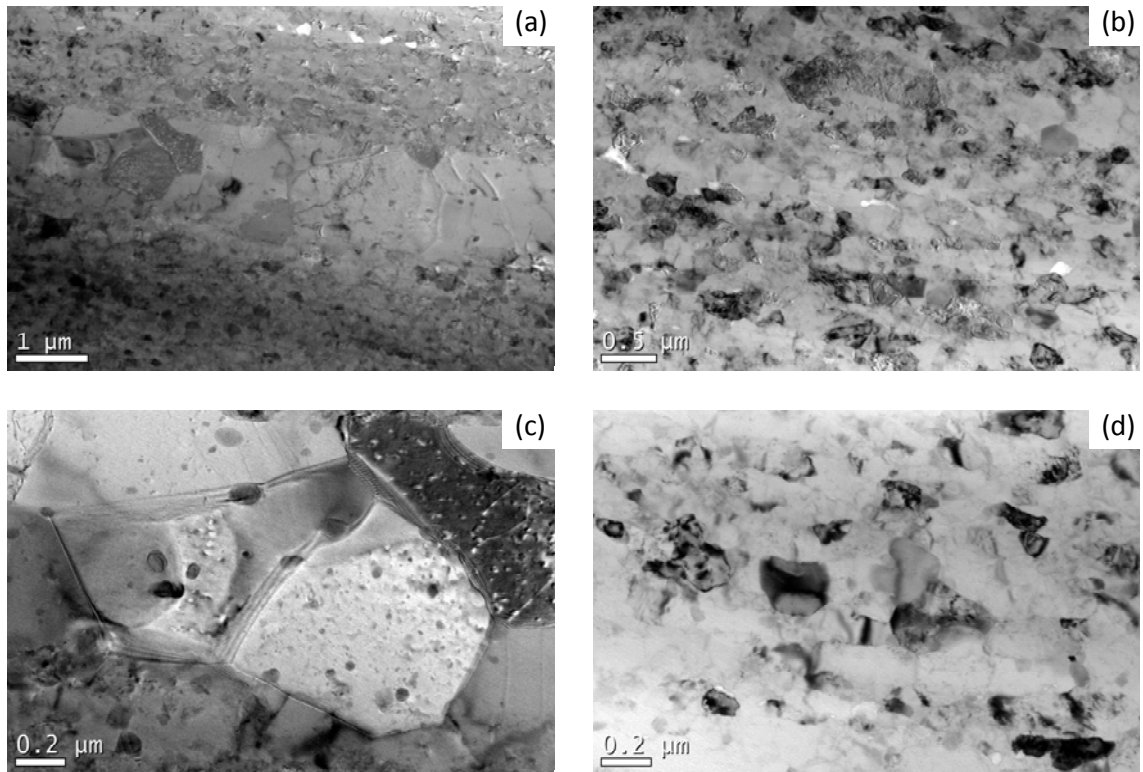


Figure 2: TEM images of CIPped and extruded Al-5083 with 10% CGs: (a) low magnitude view of CG band amid UFG matrix, (b) medium magnitude view of UFG matrix, (c) high magnitude view of CGs, and (d) high magnitude view of UFG matrix.

Since the microstructure varies considerably with fabrication process, a compilation of reported grain sizes for UFG and bimodal Al-5083 relative to the varied fabrication processes used to create them are listed in Table 2. All the information is not available in every source, but the trend is clear regardless. Ultimately the microstructure described by every source in the table falls into or around the typical characteristics of bimodal Al-5083. Exceptions include material

HIPped at the high temperature of 350°C where excessive grain growth was observed [6], and in the 50% CG bimodal samples where CGs are clustered together resulting in exceptionally large CGs and lower numbers of UFGs in the statistical set [16,18].

Table 2: Reported Microstructural Characteristics of UFG and Bimodal Al-5083

Source	CG Content (%)	Degassing Temperature (°C)	Consolidation Method, Pres. (MPa), Temp. (°C)	Deformation Method	UFG Diameter (nm)	CG Diameter (µm)
[6]	10	N/A	HIP, 150,275 HIP, 150,300 HIP, 150,325 HIP, 150,350	Extrusion	120 142 197 338	2.7 3.0 3.1 3.5
[12]	UFG 15 30	400	HIP, 7, 325	Extrusion	203 (average) 221 (average) 313 (average)	
[16,17]	50	450	CIP, 310, NA	Forging	247	4.0
[18]	UFG 15 30 50	400	CIP, 400, NA	Extrusion	207 (average) 227 (average) 317 (average) 629 (average)	
[21]	UFG 15 30	400	HIP, 172, 325	Extrusion	100 – 300	0.6 – 1.0

Material Properties

Tensile Properties

According to the Hall-Petch relation (Eq. 1), the yield strength of a material increases as the size of its grains decrease. So the constitutive properties of bimodal and UFG alloys are intricately linked to its microstructural traits. Table 3 lists the tensile properties corresponding to

the microstructures described in Table 2. To put these values in perspective, compare them to conventional polycrystalline Al-5083 which is reported to have a yield strength of 145 MPa, an ultimate tensile strength of 281 MPa, and a maximum elongation of 16% [12,17,21]. All bimodal yield strengths, exclusive of the 50% CG samples, are at least three times higher than the conventionally fabricated material. The corresponding ultimate tensile strengths are at least twice the conventional value. But every bimodal sample, including 50% CG samples, fails after straining only a third as much. The UFG samples and 15% CG content samples are even more brittle than that, straining only an eighth as much as the conventional material before failure.

Table 3: Reported Tensile Properties of UFG and Bimodal Al-5083

Source	CG Content (%)	Fabrication Summary	Yield Strength (MPa)	Ultimate Tensile Strength (MPa)	Elongation at Failure (%)
[6]	10	HIP, Extrude	640 624 611 599	N/A	N/A
[12]	UFG 15 30	HIP, Extrude	780 740 680	847 778 734	3 4 7
[16,17]	50	CIP, Forge	265	524	5.5
[18]	UFG 15 30 50	CIP, Extrude	700 650 500 400	780 770 650 570	1.5 1.8 2.1 5.2
[21]	UFG 15 30	HIP, Extrude	641 630 554	847 778 734	1.4 2.4 5.4

In addition to the overview of properties above, some of these samples have some unique behaviors than can only be described in stress-strain curves. Figure 3(a) shows compressive tests performed at different strain rates on 10% CG samples [6]. Note first that as the strain rate increases, the strength of the material decreases slightly. Second, the serrated hardening behavior that occurs at low strain rates. This is termed the Portevin-Le Chatelier effect [25] and is indicative of the occurrence of dynamic strain aging. At low strain rates, solute atoms more effectively arrest the dislocation motion and cause the stress to build for a short time before the dislocation can finally move, dropping the accumulated stress all at once. It also indicates that the deformation in the samples is highly localized.

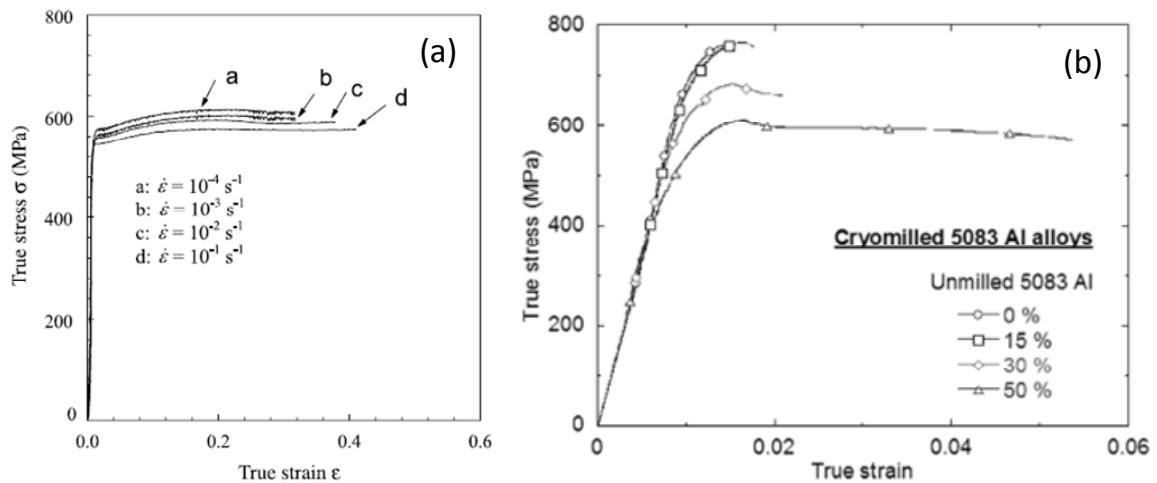


Figure 3: Stress strain curves showing (a) serrated hardening effects as a function of tensile strain rate for 10% CG sample [6], and (b) strain softening behavior of samples high CG contents [18].

Figure 3(b) shows that some bimodal metals exhibit a brief period of strain hardening after yielding, followed by a larger region of strain softening. This behavior is representative of plastic instabilities that occur once the maximum stress is reached and are associated with the formation of micro-voids and cavities and necking [18].

An all inclusive correlation between UFG grain size and strength cannot be made as the sample with the smallest grain size is not the strongest, confirming that the density and porosity levels that result from different fabrication processes play an important role in the properties of UFG and bimodal materials. No particular grouping of processes has yet been shown to be superior to any other, and the precise fabrication process one chooses for their bimodal material will be influenced by the size of sample desired, available budget, required additional forming procedures, engineering application, and possibly more.

Fatigue Properties

Fatigue is the failure of a material due to cyclic loading; that is the repeated loading of a material first in compression and then in tension (or vice-versa). The fatigue life of a metal is essentially the number of cycles of a given cyclic tension-compression load can be subjected to before failure. There are two fatigue regimes, low cycle fatigue (LCF) where the material fails in less than 1000 cycles, and high cycle fatigue (HCF) where the material fails in more than 1000 cycles.

Fatigue failure occurs in three main parts. First, micro-cracks nucleate throughout the most highly stressed regions of the material. Then, as the material is cycled further, some of the micro-cracks coalesce into a large fatigue crack that in turn propagates through the material. As the crack grows there is a point at which the reduced amount of undamaged material will not be able to withstand the cyclic load and will fail. Fatigue failures are notably ductile in nature as the material is usually deforming plastically with every cycle.

In LCF the material will endure plastic deformation at stresses above the yield point of the material. Since this is the case, research into and the models associated with LCF are directed more toward determining how the fatigue crack propagates through the material as it is cycled.

Crack initiation is assumed to occur very early in the cycling, and failure occurs when the crack length surpasses a critical value at which the material is too damaged to withstand anymore load.

Conversely, the yield strength of the material is not surpassed with each cycle in HCF. Since none of the material is being noticeably damaged, research and models associated with HCF focus heavily on crack initiation. After a large number of cycles, localized plastic deformation starts to occur [26], and after that, the same initiation and propagation process that happens in LCF occurs in HCF. Since the loads are small, the damage is incurred over a much prolonged period of cycles.

Bimodal and UFG alloys have much higher yield strengths than their conventional counterparts. This strength theoretically enables cryo-milled materials to outperform conventional metals in the HCF regime, and bimodal and UFG materials have been shown to have increased HCF lives in some experiments [14,26-28]. There is by no means a guarantee of increased performance in the HCF regime as fatigue induced grain coarsening may occur in the damaged areas, causing the crack propagation rate to be equivalent to the UFG material's polycrystalline counterparts [29,30].

In the LCF regime, there is no way to confidently predict how the behavior of UFG materials will differ from equivalent polycrystalline materials. Here the literature is conflicted. Some researchers report an increase in fatigue lives when testing UFG materials [26,27], others report a decrease [28,29], while still others describe nearly identical behaviors when comparing the same UFG and CG materials [14,31].

Literature specific to UFG Al-5083 reports on an experimental comparison between UFG, 15% CG, and polycrystalline samples. Observations made using a scanning electron microscope (SEM) show that the crack pattern differs when comparing the cryo-milled samples to polycrystalline sample. The polycrystalline Al-5083 exhibited extensive and widespread cracking across most of the sample's surface. Contrarily, the UFG and 15% CG samples had a

single dominant crack that caused failure, though the 15% CG sample did show a more diffuse cracking pattern than the UFG sample. This observation shows that the UFGs do limit the crack nucleation rate and are the source of strain localization. Interestingly, all three of the samples tested exhibited nearly identical fatigue lives [14].

Failure Mechanisms

It has been demonstrated that UFG materials are very brittle material and that CG materials are far more ductile. Combining the two in a bimodal material does not change the individual properties of each, but does create a material with better balanced macroscopic properties. Material failure could occur primarily in either microstructural region, or as a culmination of partial failures in both. The exact location of failure is very difficult to discern, especially because once the material begins to fail, a cascade effect of multiple failure mechanisms across the fracture surface occurs.

First, fracture surfaces of UFG samples of Al-5083 subjected to tensile failure are obviously brittle. Little to no necking occurs, and the fracture is jaggedly perpendicular to the loading direction. The fracture surface has many dimples and peaks whose sizes are on the order of magnitude of the UFG diameters. This indicates that the failure path primarily followed the grain boundaries [13]. On the other hand, CG samples of Al-5083 failed in tension are noticeably ductile. Significant necking and plastic elongation occur. The presence of Luder bands, which occur when dislocations are pinned by interstitial atoms and are common in Al-Mg alloys, is also noted [11,18].

Macroscopically, the failure surfaces of bimodal samples after tensile tests are oriented approximately 45° to the loading direction. Regions of necking and Luder bands are also present near the fracture surface. The most notable difference in the fracture region when compared to both the UFG and CG fracture regions are cavitations (Figure 4(a)). These cavitations appear as

dimples along the edge of the sample in the regions of largest deformation. They are hypothesized to be the result of un-accommodated strain between the UFGs and CGs [6]. Coarse grains experience substantially more plastic deformation than the UFGs do, and therefore parts of the UFG matrix must displace toward the deforming CG to maintain the cohesion of the CG boundary. This can be likened unto dislocation motion, but on an ultra-fine grain scale, with pile-ups occurring on the CG boundary. The result is the dimpled sample surface shown in Figure 4(b). Additionally, the fracture surface is an unlikely mixture of brittle UFG dimples and peaks encompassing regions of notably ductile failure of CG bands [6].

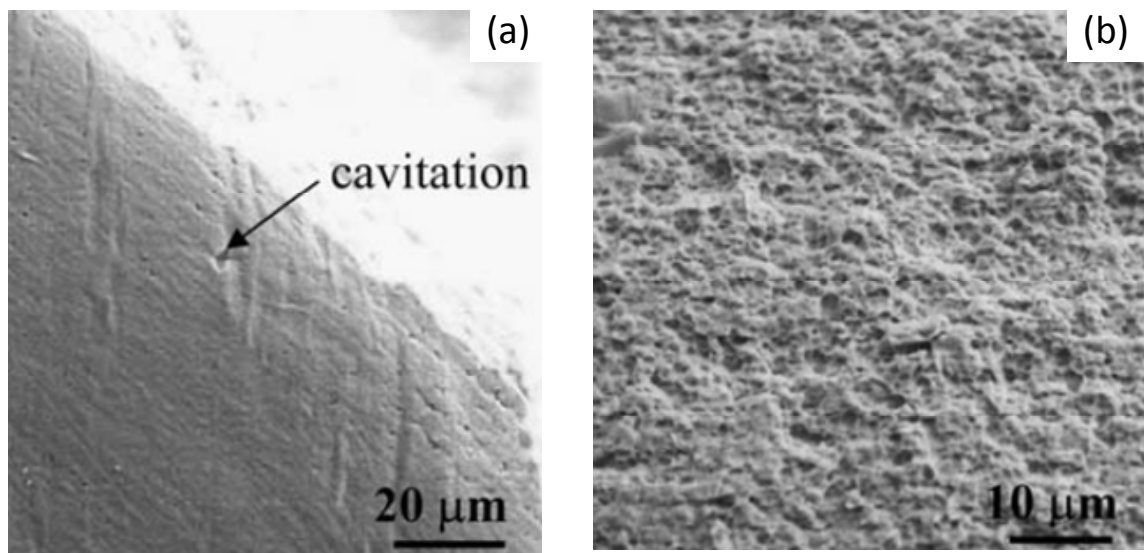


Figure 4: Fracture surface of 10% CG Al-5083 showing (a) shear fracture plane and cavitations, and (b) dimpled fracture surface.

MATERIAL MODELS

Tensile Plasticity Model for CIPped Al-5083

Tensile properties for cryo-milled, CIPped, and extruded are reported for 0%, 10%, 30%, and 50% CG contents of Al-5083 in both the longitudinal and transverse directions [11]. They are used by Joshi et al. [19] to develop a plasticity model. The linear-elastic region is modeled for all CG contents in both directions using an elastic modulus of 70 GPa and a Poisson's ratio of 0.3. Non-linear inelastic behavior is modeled using an exponential Voce hardening law, shown in Eq. 2, where σ_{pl} denotes the stress in the plastic region, σ_Y is the yield stress, R_0 , R_∞ , and b are material constants, and ε_{pl} is the inelastic strain [32].

$$\sigma_{pl} = \sigma_Y + R_0 \varepsilon_{pl} + R_\infty (1 - e^{b \varepsilon_{pl}}) \quad \text{Eq. 2}$$

It is modified to describe the hardening of bimodal Al-5083 in Eq. 3 by setting $R_0 = 0$, $R_\infty = \sigma_S - \sigma_Y$, and $b = 1/\varepsilon_c$ where σ_S is the saturation stress denoting the UTS of the alloy, and ε_c is termed the characteristic strain and is a proportionality constant [24].

$$\sigma_{pl} = \sigma_S - (\sigma_S - \sigma_Y) e^{\varepsilon_{pl}/\varepsilon_c} \quad \text{Eq. 3}$$

The Voce parameters, yield stress, saturation stress, and characteristic strain are defined as a function of CG content (f_{cg}) by Eq. 4, Eq. 5, and Eq. 6, respectively. The constants required in these equations are shown in Table 4 [24].

$$\sigma_Y = c_1 + c_2 f_{cg} \quad \text{Eq. 4}$$

$$\sigma_S = c_3 + c_4 f_{cg} \quad \text{Eq. 5}$$

$$\varepsilon_c = c_5 + c_6 f_{cg} \quad \text{Eq. 6}$$

Applying the constants shown in Table 4 to a range of CG contents yields the Voce material constants shown in Table 5. Note that this hardening law agrees with experimental results for CG contents up to 30 percent, so the properties of 100% CG material are derived independently by

Joshi, but also reported in Table 5. The stress-strain curves generated using this model for CIPped bimodal Al-5083 in the longitudinal orientation are compared to those of UFG material and 100 percent CG material in Figure 5.

Table 4: CIPped Al-5083 Voce Constants

	c_1 (MPa)	c_2 (MPa)	c_3 (MPa)	c_4 (MPa)	c_5	c_6
Longitudinal Direction	580	-218	710	-180	0.0084	-0.0086
Transverse Direction	580	-356	710	-329	0.0084	0.065

Table 5: CIPped Voce Material Parameters for a Range of CG Contents [24]

	100% CG	30% CG	20% CG	10% CG	UFG
σ_s (MPa)	425	656	674	692	710
σ_Y (MPa)	295	514.6	536.4	558.2	580
ε_c	0.035	0.00582	0.00668	0.00754	0.0084

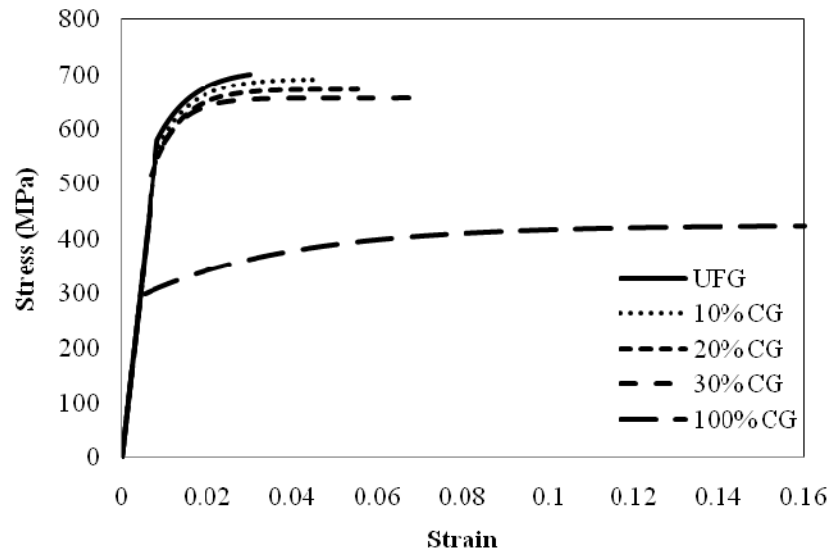


Figure 5: Stress strain curves for bimodal, UFG, and CG Al-5083 developed using Joshi's Voce hardening model.

Tensile Plasticity Model for HIPped Al-5083

Since the properties of bimodal Al-5083 vary so much with the fabrication process, and the only fatigue data presented uses HIP for the consolidation step [14], a constitutive model for HIPped Al-5083 is also developed. The tensile properties for the HIPped Al-5083 used in the fatigue experiment are reported, and are shown in Table 6.

Table 6: Tensile Properties of UFG, CG, and Bimodal Al-5083 Used in Fatigue Tests [14]

	100% CG	15% CG	UFG
σ_S (MPa)	300	450.05	482
σ_Y (MPa)	286	380.1	441
ε_{max}	0.108	0.107	0.11

Extrapolating these properties for additional CG contents and fitting the Voce hardening law to each fraction of CGs, results in constants fit for use in Eq. 4, Eq. 5, and Eq. 6 (Table 7). Applying these constants to a range of CG contents yields the Voce parameters shown in Table 8.

Table 7: HIPped Al-5083 Voce Constants

c_1 (MPa)	c_2 (MPa)	c_3 (MPa)	c_4 (MPa)	c_5	c_6
580	-218	710	-180	0.0084	-0.0086

Table 8: HIPped Voce Material Parameters for a Range of CG Contents

	100% CG	30% CG	20% CG	10% CG	UFG
σ_S (MPa)	300	418.1	439.4	460.7	482
σ_Y (MPa)	286	319.2	359.8	400.4	441
ε_c	0.01	0.006	0.007	0.00754	0.0084

And finally, the stress strain curves comparing UFG, CG, and bimodal compositions of HIPped Al-5083 are shown in Figure 6. Note that this sample of HIPped Al-5083 exhibits much more plasticity than that of the CIPped Al-5083 previously outlined. This is likely due to the variability due to processing technique where HIP or quasi-isostatic press and rolling are used.

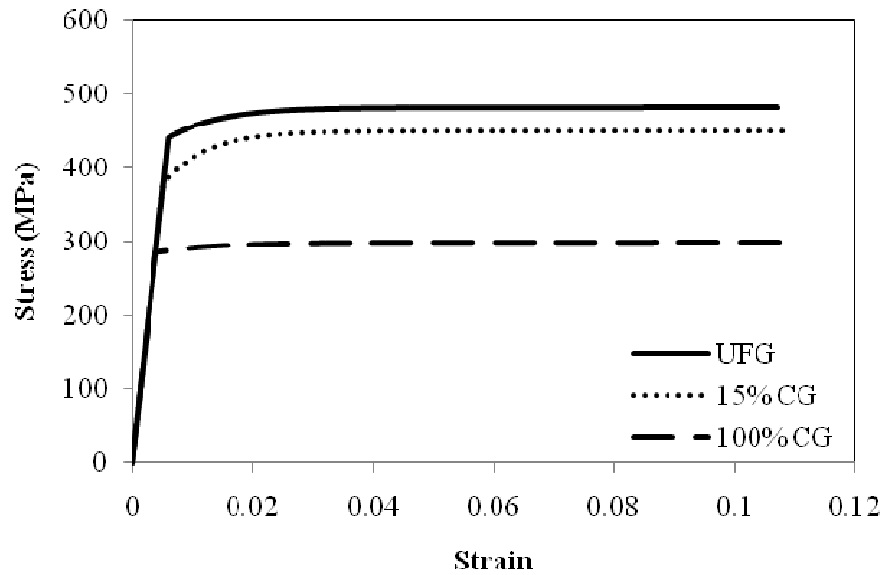


Figure 6: Tensile stress-strain curves for UFG, CG, and bimodal Al-5083 used for fatigue modeling.

Fatigue Damage Model

A low cycle fatigue (LCF) model for UFG materials was developed by Ding et al using experimental data for UFG Copper [31]. It is a crack propagation model based on the macroscopic properties of the UFG material. Since it only depends on the macroscopic properties, extending it for use on bimodal materials is reasonable, but experimental fatigue data for the material is required.

The foundation of the fatigue model is a cyclic stress strain relationship that essentially defines the material's flow stress amplitude as a function of the plastic strain amplitude applied in a fatigue scenario. This relationship is shown in Eq. 7 where K' and n' are material constant

derived by fitting a power law relationship to experimental data. By plotting the flow stress at fatigue failure for all plastic strain amplitudes tested, the curve fit, and therefore the material constants are obtained.

$$\frac{\Delta\sigma}{2} = K' \left(\frac{\Delta\varepsilon_{pl}}{2} \right)^{n'} \quad \text{Eq. 7}$$

A grain boundary strengthening and a grain boundary constraint factor are defined in Eq. 8 and Eq. 9, respectively. The first is the ratio of the UFG material yield strength, σ_{Yufg} , to the yield strength of the CG version of the same material, σ_{Ycg} , and the latter is one half the ratio of the flow stress amplitude to the effective stress amplitude.

$$F = \frac{\sigma_{Yufg}}{\sigma_{Ycg}} \quad \text{Eq. 8}$$

$$C = \frac{1}{2} \left(\frac{\frac{\Delta\sigma}{2}}{\frac{\Delta\sigma_{eff}}{2}} \right) \quad \text{Eq. 9}$$

As this is a crack propagation model it is assumed that a crack has already formed in the material. The first quantity of interest, shown in Eq. 10, is the size of the cyclic plastic zone (CPZ) around the crack tip. Here ΔK_{eff} is the stress intensity factor range, and λ is a cyclic plastic zone correction factor.

$$r_{CPZ} = \lambda \left(\frac{\Delta K_{eff}}{2\sigma_{Ycg}} \right)^2 \quad \text{Eq. 10}$$

The stress intensity factor range is defined as a function of crack length, a , and effective stress amplitude in Eq. 11.

$$\Delta K_{eff} = \frac{\Delta\sigma_{eff}}{2} \sqrt{\pi a} \quad \text{Eq. 11}$$

By solving Eq. 9 for the effective stress amplitude and using Eq. 11, the size of the CPZ is determined and shown in Eq. 12.

$$r_{CPZ} = \frac{\lambda \pi a F^2}{16 C^2} \left(\frac{K'}{\sigma_{Yufg}} \right)^2 \left(\frac{\Delta \varepsilon_{pl}}{2} \right)^{2n'} \quad \text{Eq. 12}$$

The stress field within the CPZ is defined by Eq. 13. Again, the propagation of the crack is the primary concern here, so this relation is used to define the size of the fatigue damaged zone (FDZ). Material is damaged when its local stress reaches the UTS of the material, and so it is reasonable to state that $\sigma(r_{FDZ}) = \sigma_{UTS}$.

$$\sigma(r) = \frac{\Delta \sigma}{2} \left(\frac{r_{CPZ}}{r} \right)^{\frac{n'}{n'+1}} \quad \text{Eq. 13}$$

Using the UTS of the UFG material, Eq. 7, and Eq. 12 in Eq. 13 and solving for the size of the FDZ yields Eq. 14.

$$r_{FDZ} = \frac{\lambda \pi a F^2}{16 C^2} \left(\frac{K' \frac{3n'+1}{n'}}{\sigma_{UTS} \frac{n'+1}{n'} \sigma_{Yufg}^2} \right) \left(\frac{\Delta \varepsilon_{pl}}{2} \right)^{3n'+1} \quad \text{Eq. 14}$$

Now the plastic strain field within the CPZ is defined in Eq. 15, and the accumulated plastic strain in the CPZ is defined in Eq. 16.

$$\varepsilon_{pl}(r) = \frac{\Delta \varepsilon_{pl}}{2} \left(\frac{r_{CPZ}}{r} \right)^{\frac{1}{n'+1}} \quad \text{Eq. 15}$$

$$\varepsilon_{pl}^* = \frac{1}{r_{FDZ}} \int_0^{r_{FDZ}} \varepsilon_{pl}(r) dr \quad \text{Eq. 16}$$

Using these two relations, Eq. 12, and Eq. 14, the accumulated plastic strain simplifies as shown in Eq. 17.

$$\varepsilon_{pl}^* = \left(\frac{K'}{\sigma_{UTS}} \right)^{\frac{-1}{n'}} \left(\frac{n'+1}{n'} \right) \quad \text{Eq. 17}$$

This quantity is important because it helps define the interaction energy that drives crack growth, which is defined in Eq. 18. The UTS of the UFG material is used for σ^* , and integrating Eq. 18 using cylindrical coordinates centered on the crack tip yields Eq. 19.

$$V_{\text{int}} = - \int \sigma^* \varepsilon_{\text{pl}}^* dV \quad \text{Eq. 18}$$

$$V_{\text{int}} = \pi \sigma_{\text{UTS}} \left(\frac{K'}{\sigma_{\text{UTS}}} \right)^{\frac{-1}{n'}} \left(\frac{n' + 1}{n'} \right) \left(\frac{r_{\text{FDZ}}}{2} \right)^2 \quad \text{Eq. 19}$$

In LCF, the J integral is often used to correlate crack growth [33]. It is defined in Eq. 20.

Differentiating this expression relative to the FDZ and substituting Eq. 14 results in Eq. 21.

$$\Delta J = \frac{-\partial V_{\text{int}}}{\partial r_{\text{FDZ}}} \quad \text{Eq. 20}$$

$$\Delta J = \frac{\lambda \pi^2 F^2 a}{32 C^2} \left(\frac{n' + 1}{n'} \right) \left(\frac{K'^3}{\sigma_{\text{Yufg}}^2} \right) \left(\frac{\Delta \varepsilon_{\text{pl}}}{2} \right)^{3n'+1} \quad \text{Eq. 21}$$

Finally the crack growth rate is defined in Eq. 22 in terms of the range of crack tip opening displacement (ΔCTOD), which is subsequently a function of the J integral.

$$\frac{da}{dN} = \frac{\Delta \text{CTOD}}{2} = \frac{1}{2} \left(\frac{2 \Delta J}{3 \sigma_{\text{Yufg}}} \right) \quad \text{Eq. 22}$$

Substituting Eq. 21 into this expression, separating the variables, and integrating both sides yields the Coffin-Manson relationship shown in Eq. 23.

$$N_f = \frac{96 C^2}{\lambda \pi^2 F^2} \left(\frac{n'}{n' + 1} \right) \left(\frac{\sigma_{\text{Yufg}}}{K'} \right)^3 \ln \left(\frac{a_f}{a_i} \right) \left(\frac{\Delta \varepsilon_{\text{pl}}}{2} \right)^{-(3n'+1)} \quad \text{Eq. 23}$$

Using the experimental data for UFG, 15% CG, and 100% CG Al-5083 presented by Walley et al [14], the cyclic stress strain curves and all the constants required in this model are developed and shown in Figure 7 and Table 9, respectively. The extension for other CG contents is fairly simple as it has been documented that the flow stress decreases linearly as a function of CG content [10,18,24].

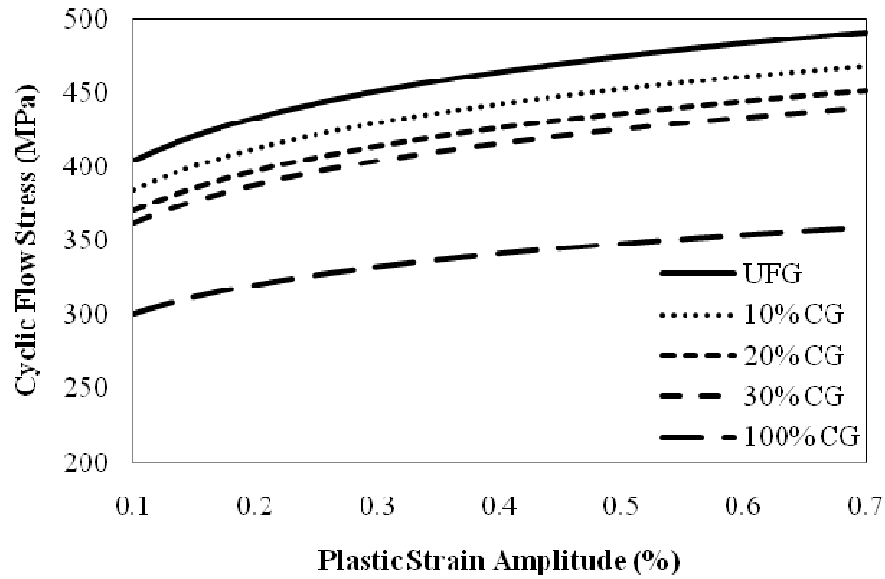


Figure 7: Cyclic stress strain curves for UFG, CG, and bimodal Al-5083.

Table 9: Fatigue Material Constants for UFG, CG, and Bimodal Al-5083

Fraction CG	UFG	10% CG	20% CG	30% CG	100% CG
$K'(MPa)$	509	485.7	467.9	455.7	370.5
n'	0.0997	0.101	0.102	0.100	0.0902
$\sigma_{Y,ufg}(MPa)$	441.0	400.3	374.5	363.4	286.0
F	1.54	1.40	1.30	1.27	1
C	0.25	0.25	0.25	0.25	0.25
$a_i (mm)$	0.0003	0.0003	0.0003	0.0003	0.0003
$a_f (mm)$	3.175	3.175	3.175	3.175	3.175
λ	0.00083	0.00083	0.00088	0.00101	0.00126

Using these relations, the fatigue lives of all the CG contents are computed and shown in Figure 8. These LCF lives agree with the experimental data [14]. It is interesting that the CG content does not affect the fatigue life in a significant way, though material comprised of purely

CGs does outperform the rest. This is not unexpected as the fatigue life of material is very much dependent upon the ductility of the material and a 100% CG sample exhibits a much higher level of ductility than any of the samples containing UFGs.

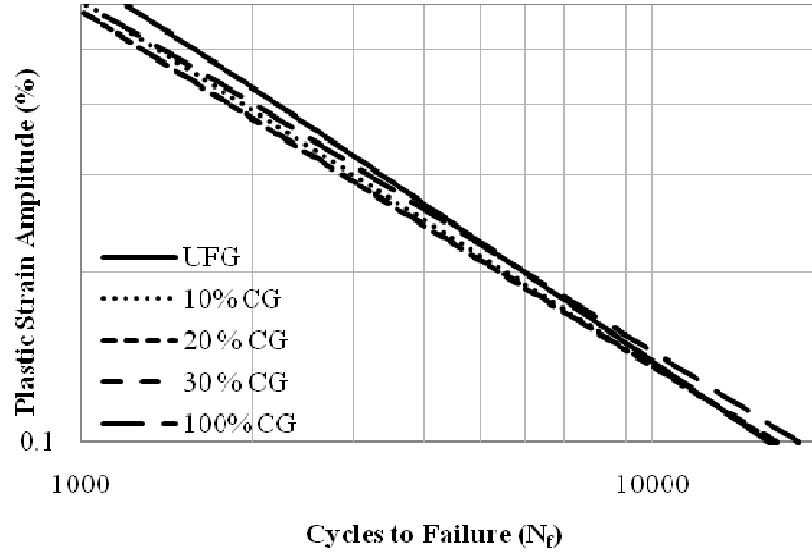


Figure 8: Coffin-Manson plot of fatigue life for UFG, CG, and bimodal Al-5083.

Since Eq. 23 is unwieldy to use in the form presented, the constants are all combined to fit a power law form shown in Eq. 24 where K is the power law constant and n is the power law exponent. The simplified constants for UFG, CG, and a range of bimodal Al-5083 are shown in Table 10. For the FE fatigue model, the UFG and 100% CG behavior are of primary interest.

$$N_f = K \left(\frac{\Delta \varepsilon_{pl}}{2} \right)^{-n} \quad \text{Eq. 24}$$

Table 10: Coffin-Manson Power Law Parameters Used to Predict Fatigue Life

	UFG	10% CG	20% CG	30% CG	100% CG
K (MPa)	290.2	232.31	211.98	212.33	228.08
n	1.444	1.544	1.587	1.589	1.599

FINITE ELEMENT MODEL

Global-Local Modeling

It is a relatively simple matter to build and solve a finite element (FE) model using constant, isotropic material properties. Although bimodal materials are not strictly isotropic and should not be modeled as such, the FE analyses being done here are uniaxial with the load axis parallel to the direction of grain elongation. Therefore, the properties in the orientation of interest can be modeled in an isotropic manner. The macroscopic material properties of bimodal Al-5083 have been investigated at length, so modeling a macroscopic tensile or fatigue scenario using FE is not difficult. But when the interactions between the CGs and the UFG matrix are desired, a macroscopic model using properties averaged from both the coarse and fine grains is inadequate. To more accurately predict the stress fields between the CGs and UFG matrix, the CGs are treated as inhomogenities that have different material properties, and difficulties arise in the size of the mesh required to model CGs.

These difficulties are resolved by using a local-global modeling technique. First, the average macroscopic properties are used to build a global model that emulates a dogbone test specimen (Figure 9), the likes of which are frequently used in tensile and fatigue testing. This model is solved either monotonically in tension, or cyclically to simulate a fatigue scenario. As shown in Fig. 6, symmetry is used to reduce the model to an eighth its total size. The mesh is biased towards the gage section since that is the primary area of interest. Multi-point constraint elements are utilized to best simulate force applied to the pinhole by a rigid pin. Both two-dimensional (2D) and three-dimensional models are developed and solved using ANSYS. For the 3D model SOLID186 elements with three translational degrees of freedom and 20 nodes are used. The 2D model uses PLANE183 elements with two translational degrees of freedom and 8 nodes.

Plasticity is modeled in both the two- and three-dimensional models using a multi-linear approach where a tabulated version of the stress-strain data for each bimodal composition is input.

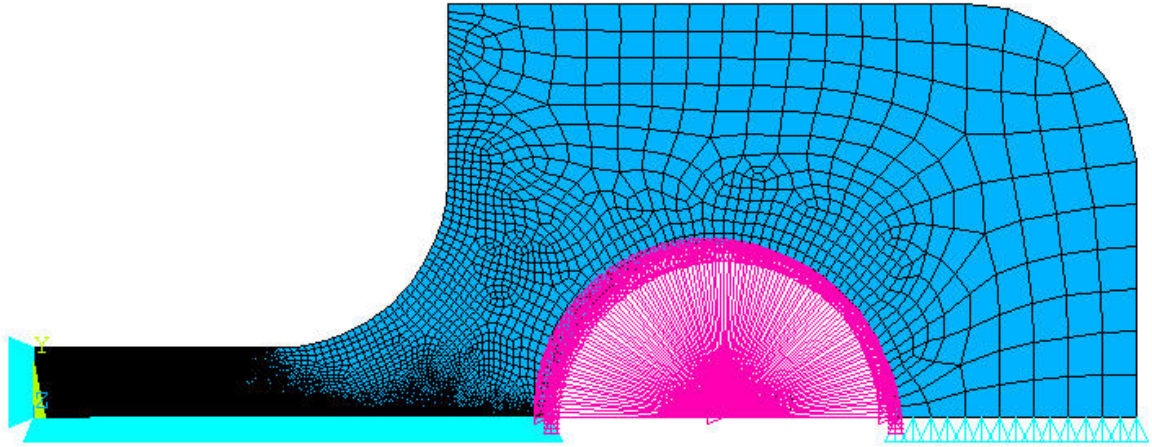


Figure 9: Global model of dogbone tensile specimen with loads and constraints described.

For monotonic loads, the applied load is incremented from the macroscopic bimodal yield strength to the bimodal macroscopic UTS. Starting the initial load at, or beneath, the macroscopic yield strength is important because in the local model the CG and UFG properties differ from the macroscopic bimodal properties. A crack could start in one region or the other when, or immediately after, the macroscopic yield strength is reached, though failure will not occur until well after this yield strength. Similarly, for cyclic loads, a load that produces a constant plastic strain amplitude is chosen, and then applied first in tension, then in compression. When solutions for each load step or each cycle are complete, the deformation from the element at the centermost portion of the model is applied to a local model that is the same size as the single element in the global model.

In the local 2D models, CGs of random sizes are randomly interspersed throughout the UFG region so that the appropriate CG ratio is attained, as shown in Figure 10. Conical coarse grains with diameters between 0.4 and 1.2 microns and lengths between 2.0 and 4.0 microns are

modeled in this random manner. Care is taken to ensure that the CGs do not overlap non-symmetric boundaries, as the model is unable to properly account for the interaction of the CGs and UFGs at the free edges.

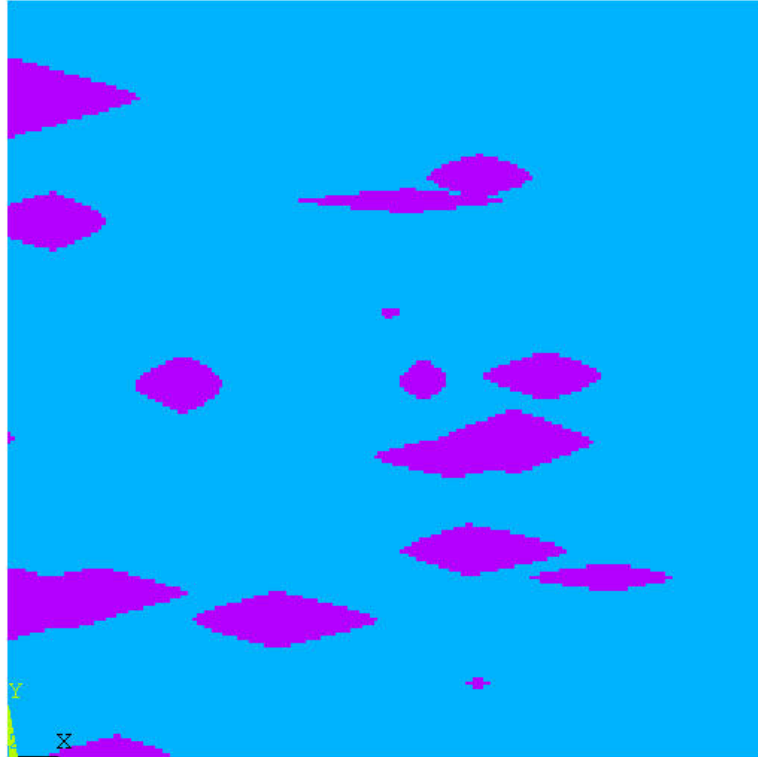


Figure 10: Local 2D model with randomly sized CGs in random locations.

When the random CG generator is used with the 3D models, the resulting menagerie of CGs proves to be exceptionally difficult to post process in a way that shows the behavior of each microstructural region and their interactions with each other. So, two large CG bands are defined instead. One long and narrow conical CG band having a diameter of 6 microns and a length of 28 microns is defined at the intersecting of all the symmetry axes, and a short and obtuse conical CG band having a diameter of 5.4 microns and a length of 12 microns is defined nearly at the center of the model, as shown in Figure 11. In this figure, only the UFG elements that make up the

model's outer edges are shown so that the locations of the CGs are easily seen relative to the UFG framework.

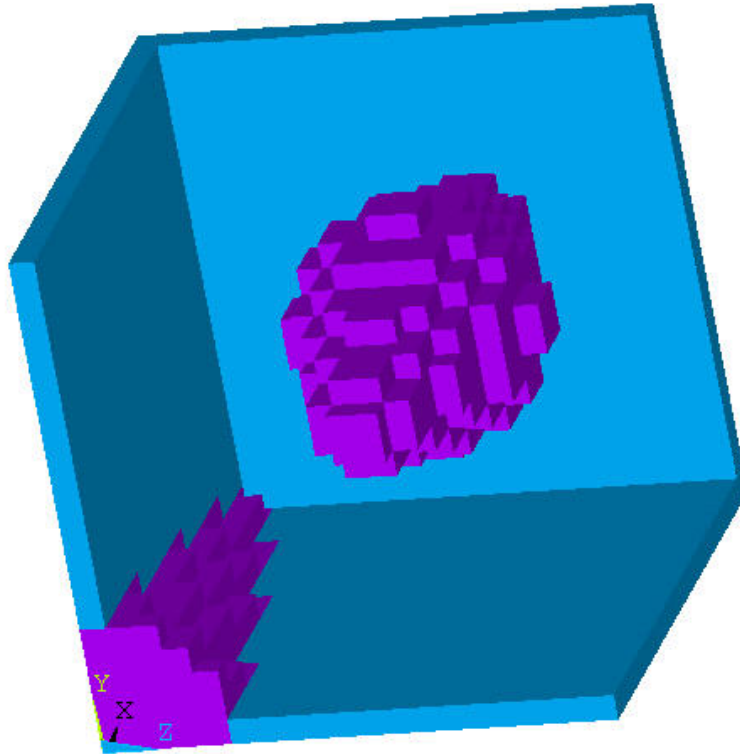


Figure 11: Local 3D model with two large CG bands: one at the symmetry intersection of the model and one at the center of the model.

Instead of using macroscopic properties defined for bimodal micro-structures, properties for UFG and CG Al-5083 are applied to appropriate regions of the model shown in Figure 10 and Figure 11. As previously alluded to, symmetry is used on all three axes. Local deformation history obtained from the global model is applied to the local model, allowing solutions be obtained. Stress, strain, and displacement fields in both the UFG and CG regions are analyzed, and elements are eliminated from the models based on either monotonic or fatigue failure criteria allowing for conclusions to be made about where in the microstructure failure occurs, and how the microstructure effects the crack propagation.

One final note about local-global modeling: In 2D, the size of the center element is small enough ($0.01\text{ mm} \times 0.01\text{ mm} \times 0.01\text{ mm}$) to apply the displacements directly from the global model to the local model. But for the 3D model, the size of the center element ($0.1\text{ mm} \times 0.1\text{ mm} \times 0.1\text{ mm}$) is still a thousand time larger than the local model can realistically be. So an intermediate model that is local compared to the global model, but still global relative to the local model, must be used. Displacements are applied to the intermediate model from the global model and from the intermediate model to the local model in the same way described above. All ANSYS input files used to build and solve the previously described models are attached in Appendix B.

Modeling Limitations

The models described above, as with any computational models, use assumptions and meshing techniques that simplify the problem being solved. First, creating finite element meshes that perfectly describe the microstructural geometry is desirable. Since the boundary between the UFG matrix and the CGs is highly irregular, using triangular (2D) or prism (3D) shaped elements to mesh the models would create a very accurate picture of the microstructure. Unfortunately, multiple difficulties arose with the meshing algorithms in ANSYS when non-rectangular elements were used. Modeling irregular shapes, like conical CGs, requires a very high resolution mesh at the CG boundaries. This resulted in far too many elements for the available computational resources to solve. Moreover, when the resolution of the mesh at the CG boundaries was forced to be within the memory limits of the computers, the coarseness of the mesh resulted in disjointed regions at the UFG:CG boundary which severely limited the usefulness of the model. Given these difficulties, the models were meshed using exclusively rectangular elements which creates a jagged interface between the CGs and the surrounding UFG matrix. But the resolutions achievable using these simpler elements and the memory limitations,

particularly in the two-dimensional models, allowed for the jaggedness of the boundary to be minimized, and the results were not adversely affected by this simplification.

Also, the properties of the UFG:CG interface are entirely unknown. But grain boundaries contribute a lot to the microstructural properties of metals as indicated by the Hall-Petch relation described previously. Fortunately, the properties of UFG:UFG boundaries and CG:CG boundaries are accounted for in the macroscopic properties of the UFG and CG regions, respectively. Only the UFG:CG interface properties remain to be determined. Since no information is available or readily attainable at this interface, its effects on the microstructure were omitted.

It would be most interesting to update the models using triangular elements, if the computational resources allow for it, and UFG:CG interface properties, once they are determined. Large deviations from the results presented forthwith are not expected, but some additional insights into the behavior of bimodal microstructures may be brought to light.

Successive Initiation

Successive initiation is a damage initiation-propagation modeling technique that has been successfully used in predicting cracks in several applications [34-38]. It involves finite element modeling in conjunction with a damage model or a failure criterion. The damage or failure criterion is applied element-wise, and failed elements are eliminated from the structure by reducing their stiffness to close to air stiffness. In monotonic modeling, the failure criteria could be yield stress, ultimate strength, yield strain, ultimate strain or total strain energy. In fatigue modeling, typically a continuum fatigue damage model is used to evaluate the accumulated damage in elements.

Successive Initiation in Monotonic Loading

The CIPped and HIPped material properties described in the Material Models section are used in the FE model to simulate monotonic loading in tension. First, the 10% CG properties are applied to the global model and it is loaded until the stress across the model's gage section is equivalent to the model's respective yield strength. Displacement values for all the nodes comprising the element in the center of the gauge length are stored. Then the load is increased for 30 steps in a way such that the stress across the model's gage length reaches the model's respective ultimate tensile strength, and the subsequent nodal displacements of the central element are also stored. All the stored displacements are in turn applied to the local model which is comprised of two distinct regions: the UFG matrix, and the CGs. After each set of displacements is applied and a solution reached, the equivalent stress in each UFG element and each CG element in the model is compared to the appropriate UTS. If the stresses in the element exceed the respective UTS, the element is eliminated from the model. Each successive load step increases the number of eliminated elements, thereby propagating the path of tensile fracture.

Successive Initiation in Cyclic Loading

Successive initiation in the fatigue analyses require several steps. The damage initiation site is first identified with the help of a Coffin-Manson damage model which predicts the cycles to failure as a function of plastic strain amplitude. After a single cycle of the model, the plastic strain amplitude in every element is known, and the number of cycles to failure throughout the model is determined. All elements that have fatigue lives less than a threshold value are eliminated and identified as the damage initiation zone. Damage is accumulated in every other element throughout the model according to Eq. 25 where D^* indicates the accumulated damage after the current load step, D_0 represents damage accumulated in previous load steps, N_f is the number of cycles to failure predicted in the current load step, and N_{cyc} denotes the number of

cycles that the current load step simulates. The accumulated damage is stored and added to with each successive simulated FE cycle. Whenever the accumulated damage reaches unity, the element in question is assumed to have failed and eliminated.

$$D^* = D_0 + N_{cyc} \frac{1}{N_f} \quad \text{Eq. 25}$$

The HIPped fatigue properties of Al-5083 and the corresponding constitutive properties used in the fatigue experiments are described in the Material Models section. The global and local models are set up in an identical manner as the models for the tensile simulations where the global model used the 10% CG material properties, and the local model has regions of CGs distributed throughout the UFG matrix. First, the global model is cycled in tension and compression in such a way as to produce a plastic strain amplitude of 0.1%. Nodal displacements of the centermost element are saved for both the tension and compression steps. These displacements are applied to every cycle of the local model. After each cycle in tension and compression, the plastic strain experienced by every element is stored. The tensile and compressive strains are averaged together to provide the plastic strain amplitude that every element experiences under the loading conditions. Using Eq. 24 with the appropriate UFG or CG properties to predict the fatigue lives of every element after each load step allows the successive initiation criteria described in Eq. 25 to be used to determine the state of damage in every element so that only a fraction of the total cycles need be solved. The crack is initiated in the first step by eliminating the elements with exceptionally short fatigue lives. Damage in the rest of the elements is accumulated based on the fatigue lives of the eliminated elements (N_{cyc}) and the fatigue lives of each remaining element (N_f). As the model is cycled further, elements are eliminated when their value of accumulated damage reaches or exceeds unity using 200 cycles per FE iteration.

FINITE ELEMENT RESULTS

This section contains many microstructural FE figures, all of which are color coded in the same manner. Ultra-fine grain regions are shown in blue and coarse grain regions in purple. Red failure paths indicate eliminated UFG elements, and yellow-green failure paths denote eliminated CG elements. This color scheme applies to both the 2D and 3D models. Also, it is difficult to illustrate the path of crack propagation in the 3D models using 2D figures. So, for the sake of visual clarity, the failed elements and the CG elements are shown framed by only the outer edge of the model which is comprised of almost entirely UFG elements. Though only the outer edge UFG elements are shown, all the empty space in the figures is occupied by live UFG elements.

Furthermore, all the results presented here are the highest resolution possible given the computational power of the machines being used. For the 2D models, a grid size of 200×200 is used, making the size of each element in the models $50 \times 50 \text{ nm}^2$. The 3D models have grid sizes of $20 \times 20 \times 20$, or individual element sizes of $500 \times 500 \times 500 \text{ nm}^3$. When compared to results previously obtained using 2D resolutions of 100×100 and 3D resolutions of $15 \times 15 \times 15$, the site of crack initiation and the path of propagation do not change (see Appendix A). Cracks formed a load step earlier and propagated a load step faster in the high resolution models, particularly when loaded cyclically. This is attributed to the smaller elements on the grain boundaries being subjected to a larger portion of the stress concentrations located there. Since the crack nucleation sites and propagation paths are of primary interest in this study, it is reasonable to conclude that no more grid refinement is required.

Tensile Failure of CIPped Al-5083

Two-Dimensional Model

To model the tensile failure in two dimensions loads are applied to the global model starting at the macroscopic yield strength (560 MPa) and ending at the macroscopic ultimate strength (690

MPa) of bimodal Al-5083 comprised of 10 percent CGs. Fifteen steps are solved, each accounting for 11.7 MPa of sequential loading.

Cracks initiate after one load step when the central gage stress in the global model is 560 MPa. They are shown in Figure 12(a) as single failed UFG elements on the boundaries of CGs. Higher magnitude views of the failed elements are shown in Figure 12(c) and Figure 12(d), which

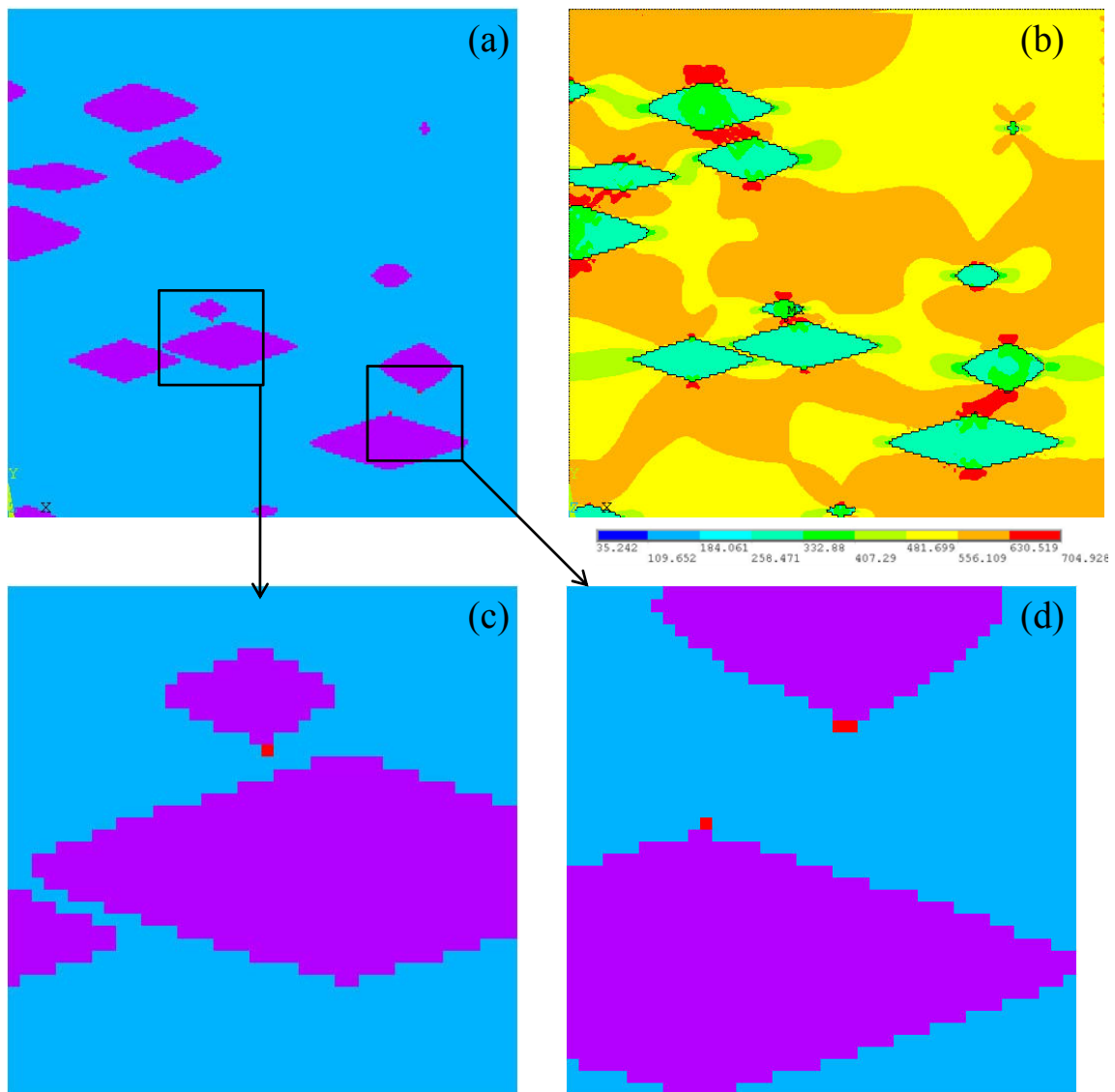


Figure 12: Crack initiation in tension corresponding to a gage section stress of 560 MPa (load step 1) in the global model; (a) eliminated elements in the microstructure, (b) stress fields in the UFG matrix and CG regions, (c) and (d) enlarged views of the areas

correspond to the boxed regions in Figure 12(a). Stresses throughout the local model are shown in Figure 12(b), and it is immediately apparent that the high stress regions are localized to the UFG matrix near the CG boundaries. Many of these regions are very near the UTS (690 MPa) of the UFG elements, indicating that failure could be imminent in all of these high stress areas. Also note that the stresses in the CGs are well below the CG ultimate tensile strength of 390 MPa. The lower stress in the CGs is directly attributed to their higher level of ductility. In other words, given equivalent strains in the UFG and CG elements, the corresponding stress level in the UFGs is much closer to their ultimate strength than the stress level in the CGs.

Two load steps later, the cracks grow in the UFG matrix, extending towards the nearest neighboring CGs, as shown in Figure 13(a). They also breach the boundaries into the CGs they started next to. Also, another crack forms on the opposite side of the most heavily affected CG. After the the fourth load step the growing crack is shown completing its passage through the CG and extending into the UFG matrix in Figure 13(b). Also note that the second crack has completed its progression between its two nearest CGs. Two steps later the crack extension is almost complete, as shown in Figure 13(c). The two cracks have moved toward each other, through every CG they were in contact with, as well as one that was previously unaffected, and extend through the UFG matrix to the boundaries of the model. The model fails completely six steps later in Figure 13(d) when the stress in the global model's gage section is 690 MPa, the global ultimate strength.

In summary, the crack starts at a boundary between the UFG matrix and a CG. This occurs because the brittle ultra-fine grains cannot withstand deformation as well as the ductile coarse grains. Once the crack is started, the surrounding UFGs fail very soon, and the stress concentrations at the crack tip that is pressed up against a CG cause said CG to fail. This process repeats itself as the crack propagates through the UFG matrix towards surrounding CGs, swallowing them up as soon as contact is made.

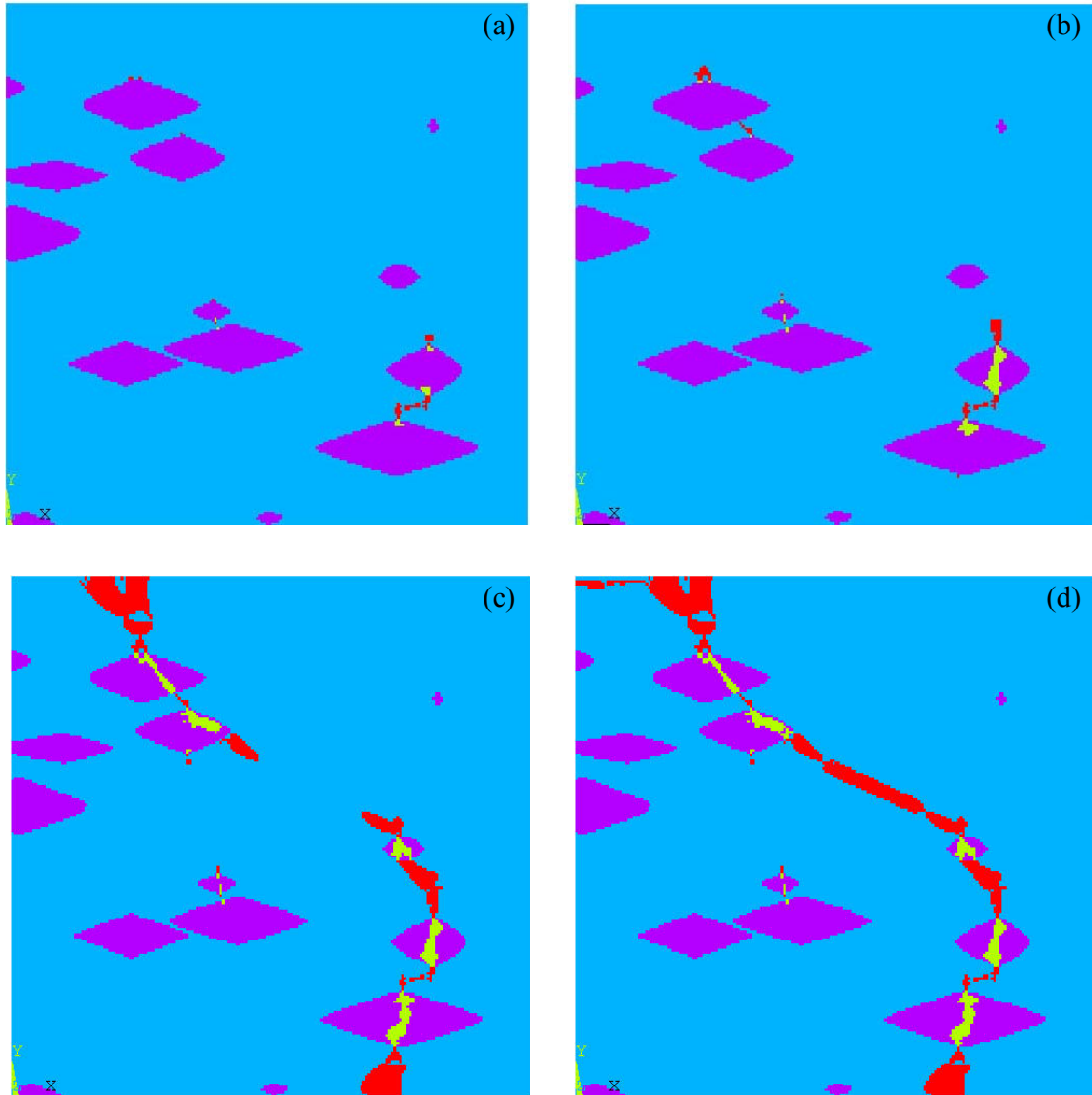


Figure 13: Crack propagation in tension corresponding to a gage section stress of (a) 583 MPa (load step 3), (b) 595 MPa (load step 4), (c) 618 MPa (load step 6), and (d) 688 MPa (load step 12) in the global model.

Three-Dimensional Model

The three dimensional successive initiation solutions are determined by increasing the load for 30 steps from the global yield strength (560 MPa) to the global ultimate strength (690 MPa) of bimodal Al-5083 comprised of ten percent CGs. This results in a step size of 4.3 MPa applied sequentially.

Similarly to the 2D model, the crack initiates in the UFG matrix at the boundary of a CG, as shown in Figure 14(a). This figure, and all following figures in the 3D analyses, show the failed elements and the CGs framed by the edges of the UFG matrix. A cross sectional slice of the model showing the stress distribution in the UFG matrix surrounding the initial crack is shown in Figure 14(b). Coarse grain regions are omitted to provide a clear picture of the stress in the UFG matrix. The stresses in the CGs are far lower than their ultimate strength and do not largely impact the stresses at the location of crack initiation, therefore they are not shown here. Stress levels in the UFGs increase with their proximity to the CG, peaking at the boundary, particularly where the failure starts. As stated in previously in the 2D model, UFG stresses at the interface are significantly higher because the interactions at the boundary cause the UFG region to displace more than it would in the absence of the CG. This increase in strain corresponds to a higher level of stress, and ultimately causes the UFG material to fail at the boundary.

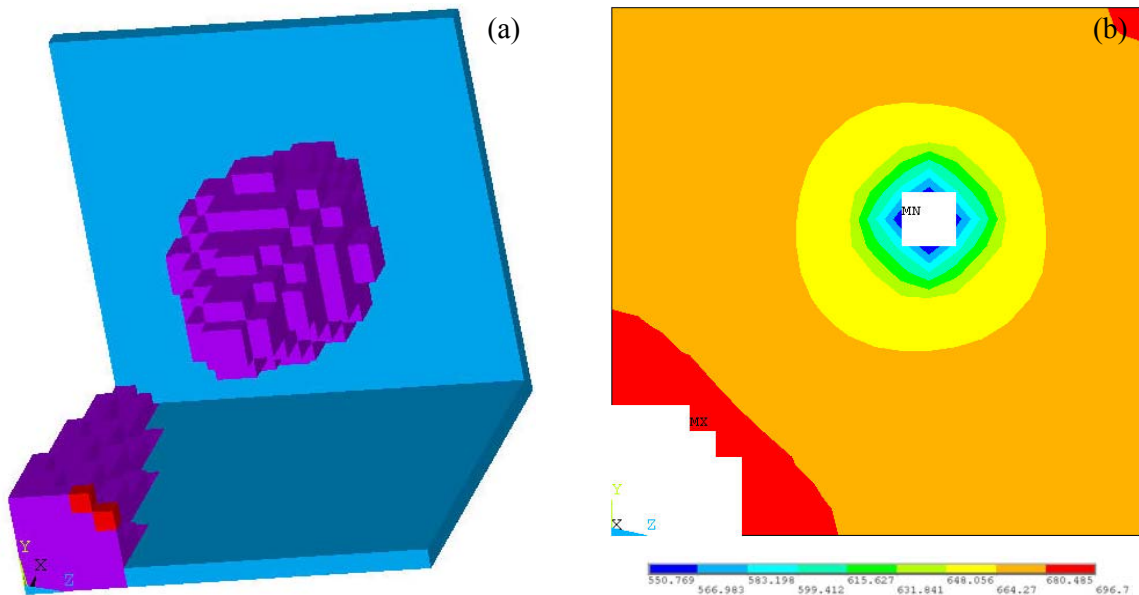


Figure 14: Crack initiation in tension corresponding to a gage section stress of 595 MPa (load step 9) in the global model; (a) eliminated elements in the microstructure, (b) stress fields in a UFG matrix cross section taken at the x-location of the initial crack.

Four load steps later, more UFG elements surrounding the site of initiation have failed, as have elements surrounding the other CG at the symmetry corner of the model (Figure 15(a)). Ultra-fine grain elements between the two initiation sites also fail, indicating the initial path of propagation. This is further illustrated an additional step later by Figure 15(b) where the

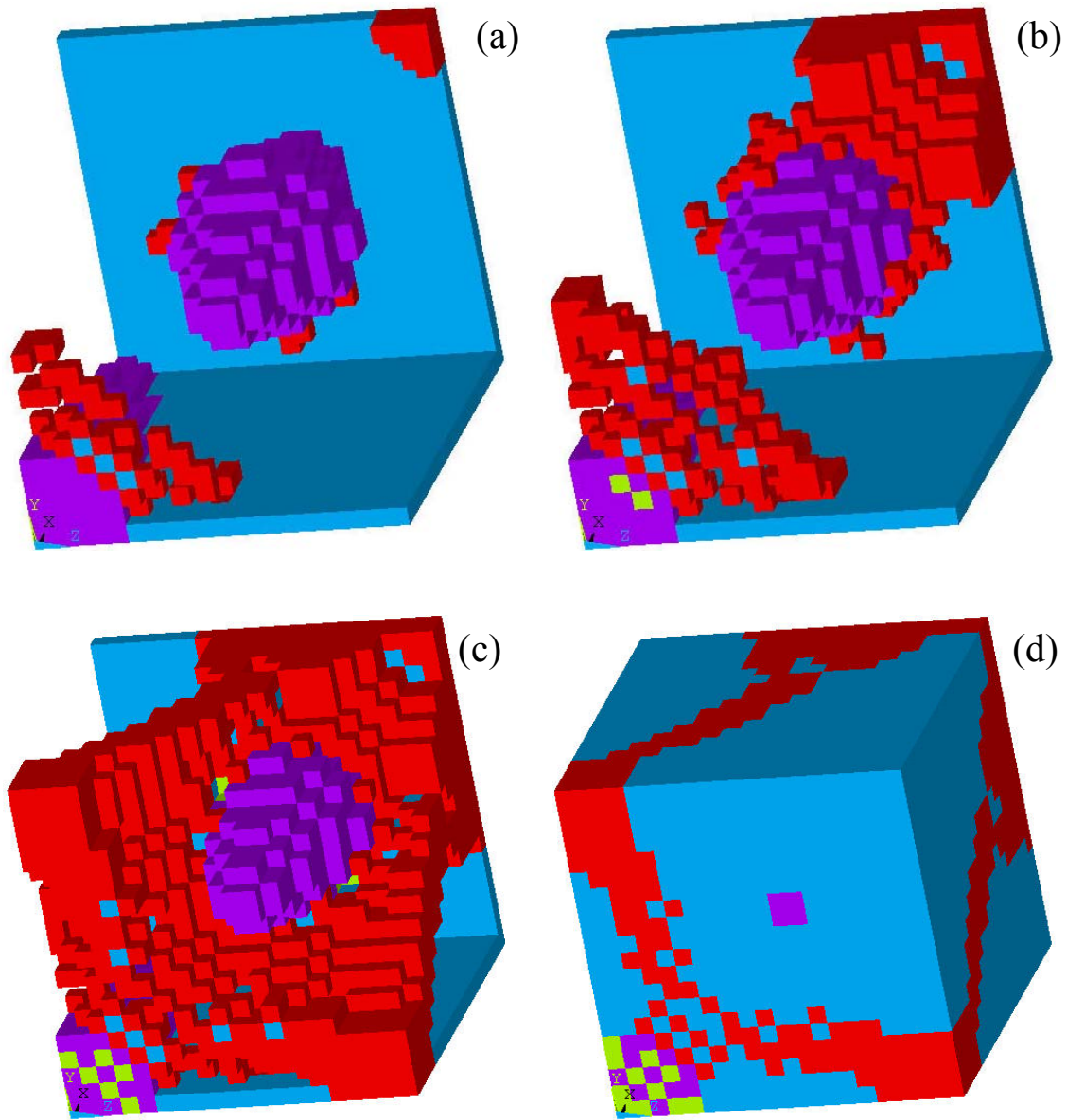


Figure 15: Crack propagation in tension corresponding to gage section stresses of (a) 612 MPa (load step 13), (b) 616 MPa (load step 14), (c) 625 MPa (load step 16), and (d) 690 MPa (load step 30) in the global model.

connection between the CGs is almost complete. The crack has penetrated the CG on the symmetry boundaries, but unseen in these figures is the failure of elements in the center CG, which is also occurring.

Two steps later, the crack spans the length of the model as shown in Figure 15(c), and the affliction to the CGs is finally visible, particularly in the CG on the symmetry boundaries. At this point, the model is as good as failed, but through the course of 15 more steps, small amounts of additional failure occur, as shown in Figure 15(d), before the model is unable to solve anything further.

The two and three dimensional models agree very well in that crack initiation occurs in the UFG matrix on the boundary of a CG. Furthermore, there are multiple points of nucleation after initiation. In both cases the cracks initially move through the UFG matrix before afflicting the CG regions, and once the CGs are affected, both models show that they fail very quickly thereafter. Failure is attributed to the brittleness of the UFGs. Given the same displacements, they will reach their ultimate strength much sooner and fail. The crack that results from their failure creates zones of concentrated stress that cause a cascade effect on both the surrounding UFGs and CGs.

Tensile Failure of HIPped Al-5083

The tensile properties described for HIPped UFG and bimodal Al-5083 are primarily included as they pertain to the fatigue model. They are by no means representative of the majority of cryo-milled Al-5083 since the fabrication process used (HIP + forging) reportedly resulted in UFG and bimodal samples that are four times more ductile than typical samples. Additionally, the reported strengths of these samples are relatively low when compared to most cryo-milled Al-5083. Nonetheless, the tensile properties are used to predict the crack nucleation sites and propagation path in the same manner used for the more typical CIPped material. In both the 2D and 3D

models, the loads are applied to the global model starting at the macroscopic yield strength (380 MPa) and ending at the macroscopic ultimate strength (450 MPa) of bimodal Al-5083 comprised of ten percent CGs. Ten steps are solved, each accounting for 7 MPa of sequential loading.

Two-Dimensional Model

A crack nucleates when the stress in the global model is 387 MPa (load step 2), which is uncharacteristically close to the macroscopic yield stress, as shown in Figure 16(d). It initiates in the CG and quickly spreads to the UFG matrix in the next step (394 MPa) shown in Figure 16(c).

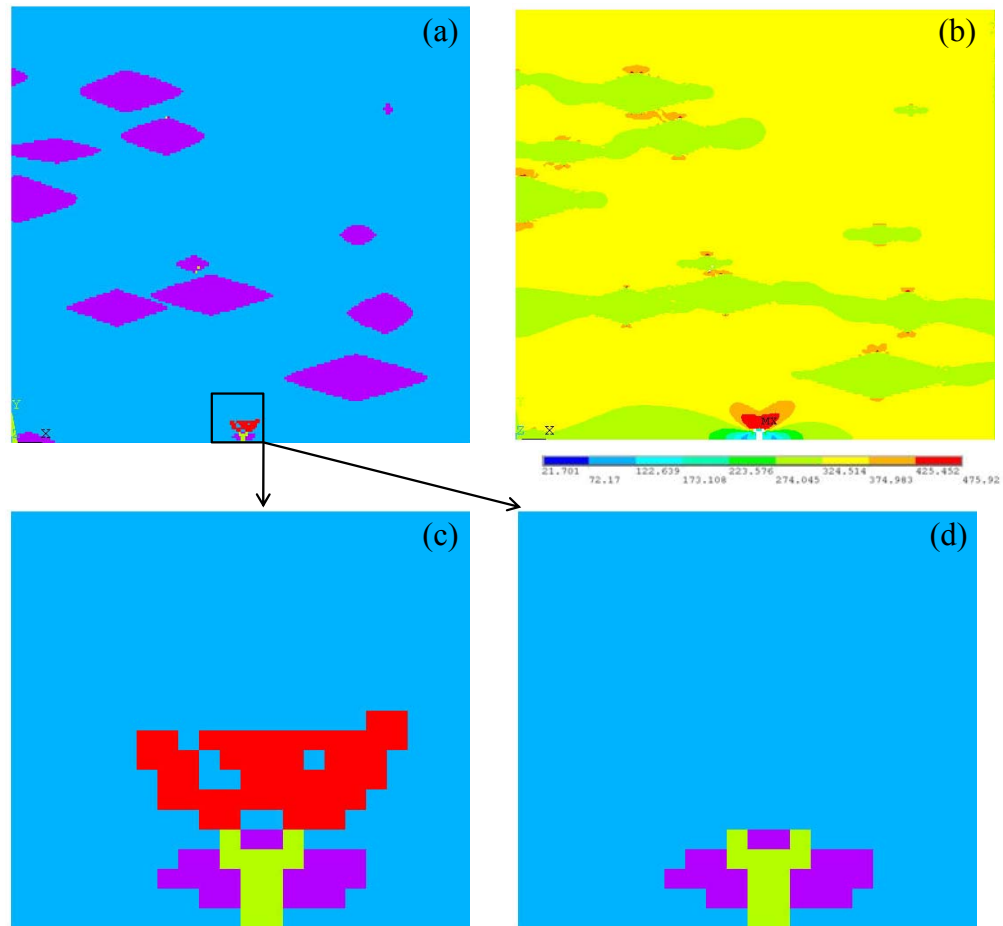


Figure 16: Crack initiation in tension corresponding to a gage section stress of 387 – 394 MPa (load steps 2 – 3) in the global model; (a) eliminated elements in the microstructure, (b) stress fields in the UFG matrix and CG regions, both after the third load step, and (c) and (d) enlarged views of the areas indicated in (a) after the third and second load steps, respectively.

Both these illustrations are magnified views of the full model shown in Figure 16(a). Stresses in the model are illustrated after the third step in Figure 16(b), where it is apparent why failure occurs in the region it does. High stresses in the UFG regions between CGs are also visible, and additional nucleation sites are likely.

As expected, a load step later (401 MPa) additional nucleation sites are visible (Figure 17(a)). Also, the initial crack is propagating through the UFG matrix towards the nearest CG. The next

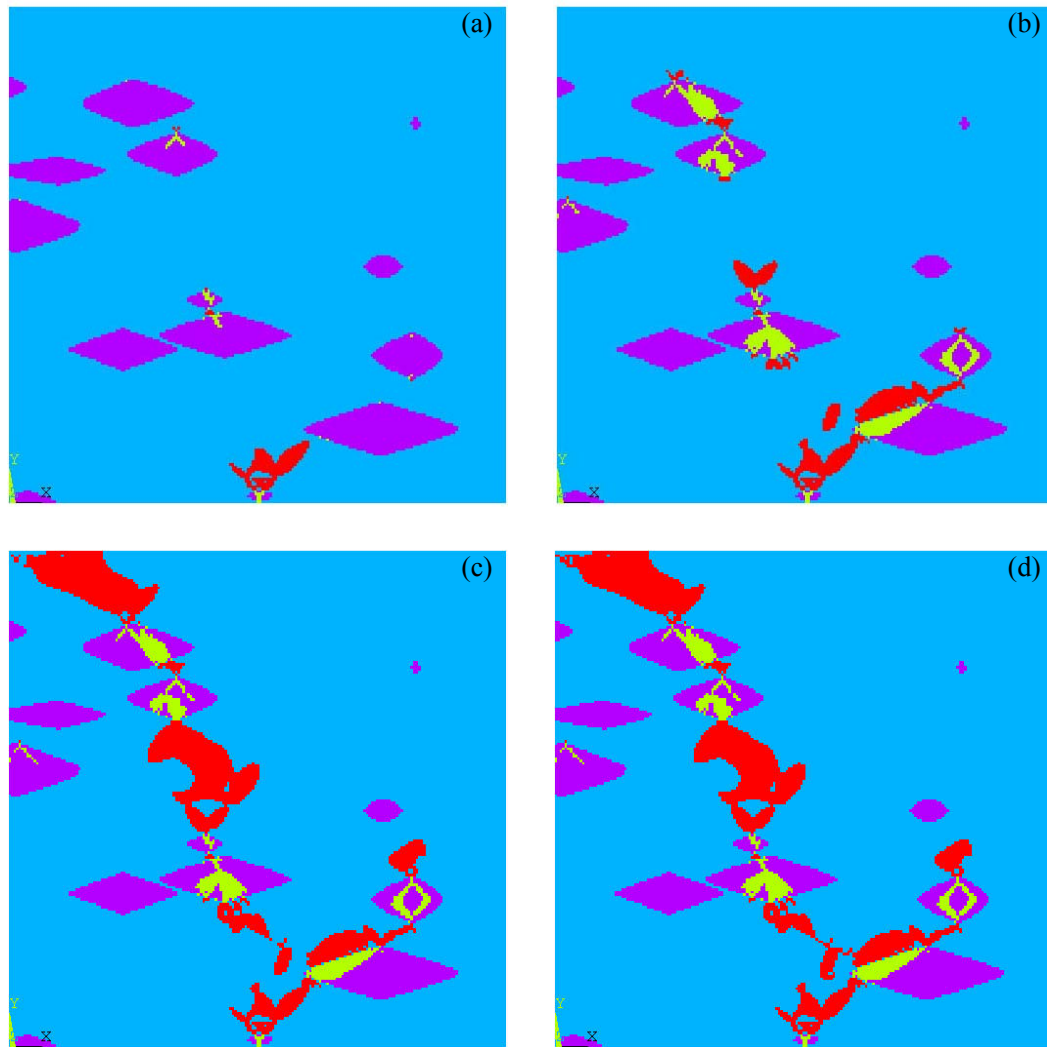


Figure 17: Crack propagation in tension corresponding to a gage section stress of (a) 401 MPa (load step 4), (b) 408 MPa (load step 5), (c) 415 MPa (load step 6), and (d) 422 MPa (load step 7) in the global model.

three load steps (408, 415, and 422 MPa respectively) are shown in Figure 17(b-d), respectively. At 408 MPa of global load both the primary and secondary initiation sites are expanding quickly. They coalesce at 415 MPa of global load immediately before the model fails at 422 MPa.

When compared to the CIPped FE results, these HIPped FE results are notably different. First, the CIPped model fails at strengths much closer to the global ultimate tensile strength. And second, the crack nucleates within the UFG matrix on the boundary of a CG in the CIPped model, but the first elements to fail in the HIPped model are CG elements. The first of these differences is attributed to the curve fit applied to the tensile properties presented alongside the experimental fatigue results. Since there is no way to know the precise hardening behavior, the curve fitting procedure used for the CIPped materials is used, and may not be truly representative of the HIPped material. Additionally, because the HIPped material described has properties that are unrepresentative of most cryo-milled Al-5083 (relatively low strengths, and exceptionally high ductility) the applied deformations affects both the UFG and CG regions in almost the same way. The uncharacteristically high elongations at failure (approximately 10% for CG, UFG, and bimodal samples) explain the difference in crack nucleation sites as well. Since both the UFG and CG regions fail after experiencing about the same level of deformation, it is impossible to predict which one will fail first. In this 2D case, it is the CGs, where in both the CIPped cases it is the UFGs. The 3D HIPped case is described forthwith, and the different site of crack initiation verifies this unpredictability due to the uncharacteristically ductile HIPped material being modeled.

Three-Dimensional Model

In this model, the crack initiates in the UFG matrix as expected (Figure 18(a)) during the first load step (380 MPa of global stress). Nucleation sites appear on both CG boundaries and correspond to the stress distribution shown in Figure 18(b). This result, surprisingly, agrees more

with the 3D CIPped initiation location and less with the 2D HIPped initiation location just presented.

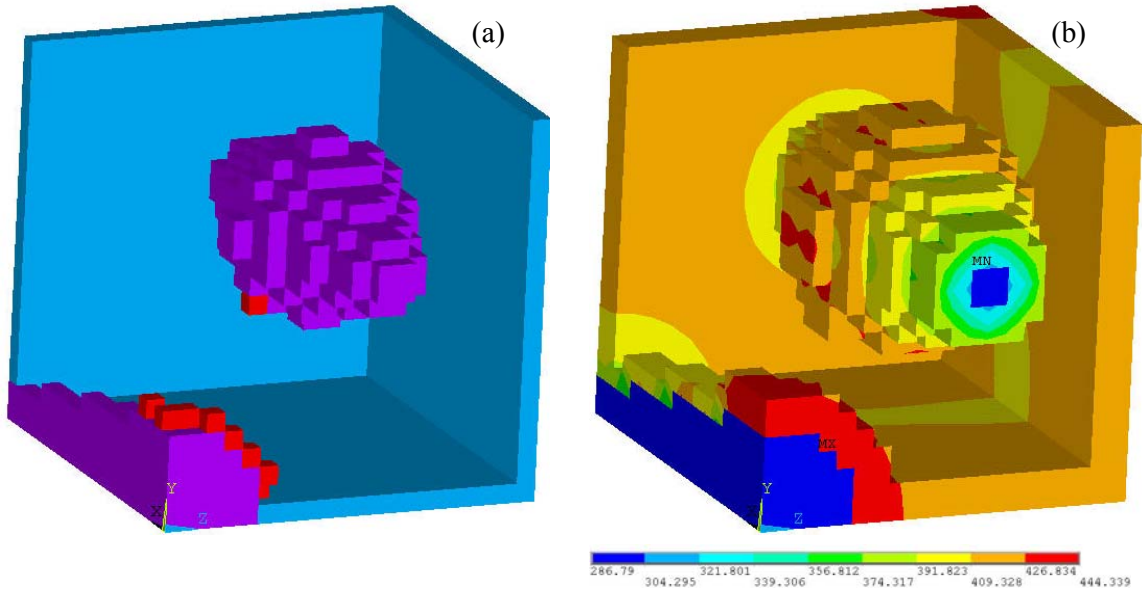


Figure 18: Crack initiation in tension corresponding to a gage section stress of 380 MPa (load step 1) in the global model; (a) eliminated elements in the microstructure, (b) stress fields in the UFG matrix relative to CG locations.

The following load step (387 MPa of global stress) shows substantial crack propagation through the UFG matrix (Figure 19(a)). Figure 19(b) shows the next load step (394 MPa) where the crack has propagated through the CG on the symmetry axes, but has not yet affected the central CG. It has surrounded said CG, but it takes one more load step (401 MPa) for it to fail as shown in Figure 19(c) where the live CG elements have been omitted so the extent of failure in the CGs can be more readily seen. Figure 19(d) illustrates the path of failure in the full model after the same final load step.

As mentioned in the 2D HIPped results, these 3D HIPped results are more in line with the 3D CIPped results. The crack nucleates in the UFG matrix and propagates between the CGs before penetrating them. Ultimately, the only real agreement this 3D model has with the preceding 2D

model is the accelerate rate of failure. Both models fail much sooner that the CIPped models due, relative to the global ultimate tensile strengths. The explanation given at the end of the 2D HIPped model is valid here as well.

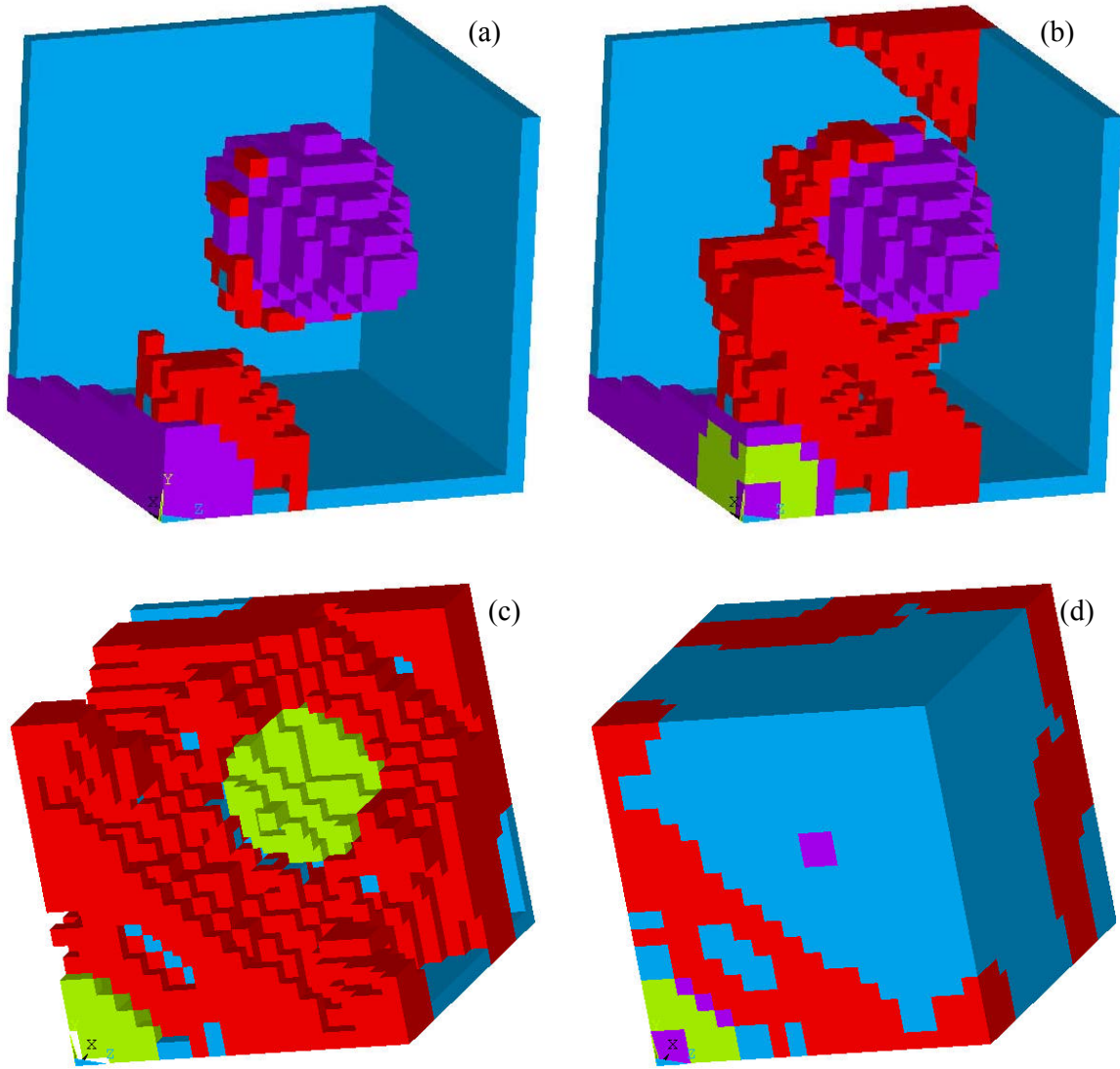


Figure 19: Crack propagation in tension corresponding to gage section stresses of (a) 387 MPa (load step 2), (b) 394 MPa (load step 3), and (c – d) 401 MPa (load step 4): (c) interiors of CGs and (d) full model after failure.

Cyclic Failure of HIPped Al-5083

The successive initiation solution for a cyclic loading scenario is determined using tensile and compressive displacements obtained from the global model given a plastic strain amplitude of 0.01 and applying them to the local model. Each FE iteration simulates 25 cycles (2D model) or 50 cycles (3D) of fatigue loading with damage accumulated in an element by element basis. All elements start with an initial damage of zero, and are considered to have failed when their damage value equals or exceeds unity.

Two-Dimensional Model

Initially, fatigue micro-cracks nucleate within the CGs where they intersect with the UFG matrix after 100 simulated cycles, as shown in Figure 20(a). Higher magnitude views of the failed regions are shown in Figure 20(c) and Figure 20(d) and correspond to the boxed areas of Figure 20(a). Most of the model remains undamaged through the first iteration, but isolated areas corresponding with the CG boundaries are seriously afflicted by damage. This observation is in direct contradiction to the monotonic results previously presented, but is not unexpected. When subjected to equivalent global plastic strain amplitudes, the CG regions experience higher levels of plastic deformation than the UFG regions. Since the UFG and CG damage models do not differ in a large way, the higher plastic strain amplitude the CGs endure reduce their fatigue lives significantly therefore increasing the damage they accumulate to the point of failure.

Note that in Figures 20 (c) and (d), there are a few UFG elements that are damaged enough to fail. These elements have accumulated far lower amounts of damage when compared to the failed CG elements. In other words, the UFG elements do not fail at the same time as the CG elements, they fail as a result of the CG elements around them failing. The subsequent finite element iteration after 125 cycles have been simulated is shown in Figure 20(b). This figure

shows how the crack initially propagates through the CGs without affecting the UFG regions at all.

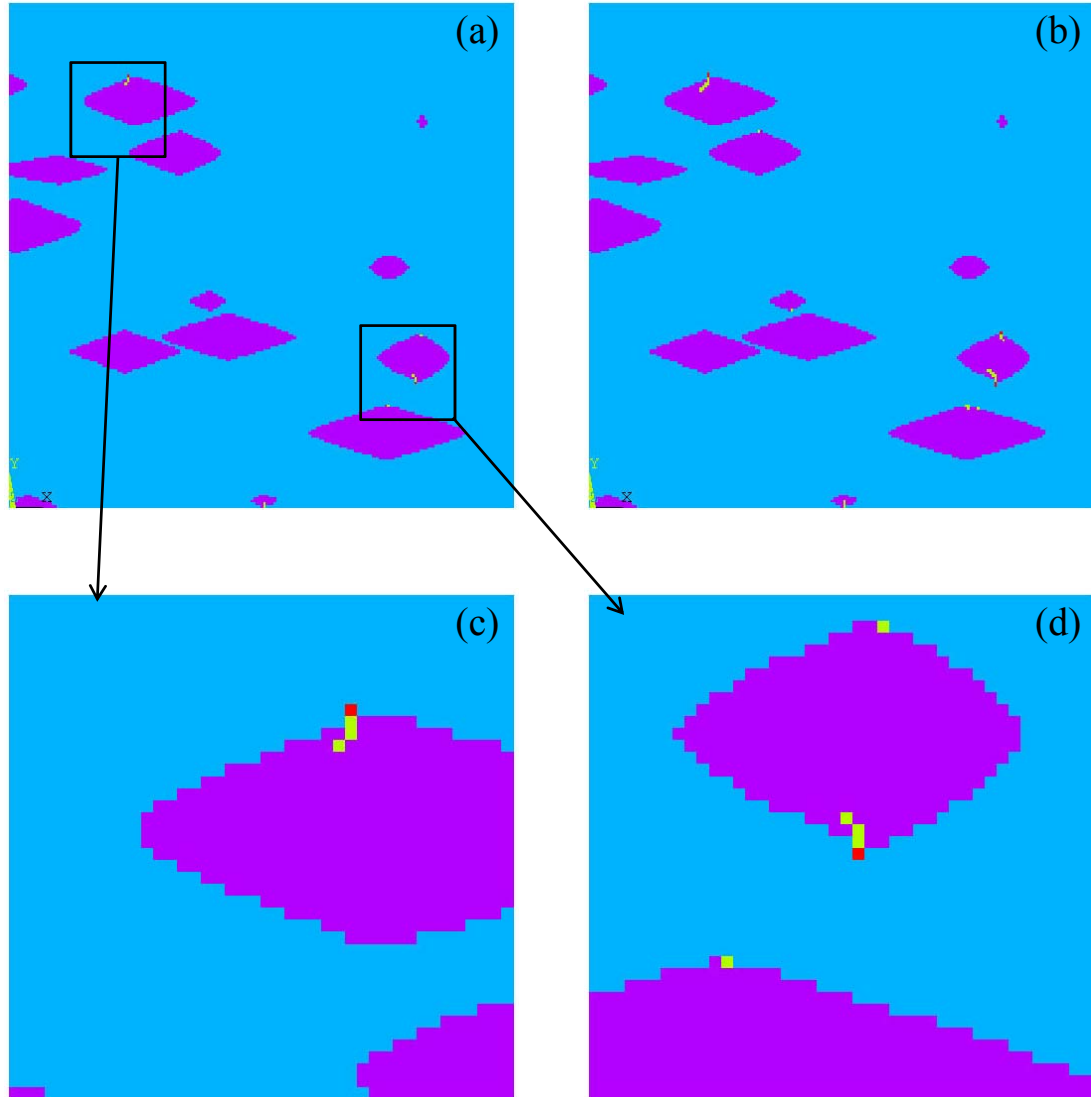


Figure 20: Crack initiation and propagation in cyclic loading after (a) 100 simulated cycles, and (b) 125 simulated cycles. Enlarged views, (c) and (d), of areas indicated in (a).

After 25 additional simulated cycles (150 total), the multiple nucleated micro-cracks begin to coalesce into fracture systems (Figure 21(a)). Coarse grains subjected to high levels of damage have cracked through. As the cycling continues, cracks form between affected CGs in close

proximity to each other, quickly spanning the UFG matrix in between (Figure 21(b)). When the simulated cycles reach 200, the crack begins to span the larger gaps between damaged CGs, as shown in Figure 21(c). Failure occurs after 325 simulated cycles when the crack spans the entire model (Figure 21(d)). The crack propagates through the remaining UFG matrix, taking the path of least resistance and connecting through the CGs closest in proximity to the crack tips.

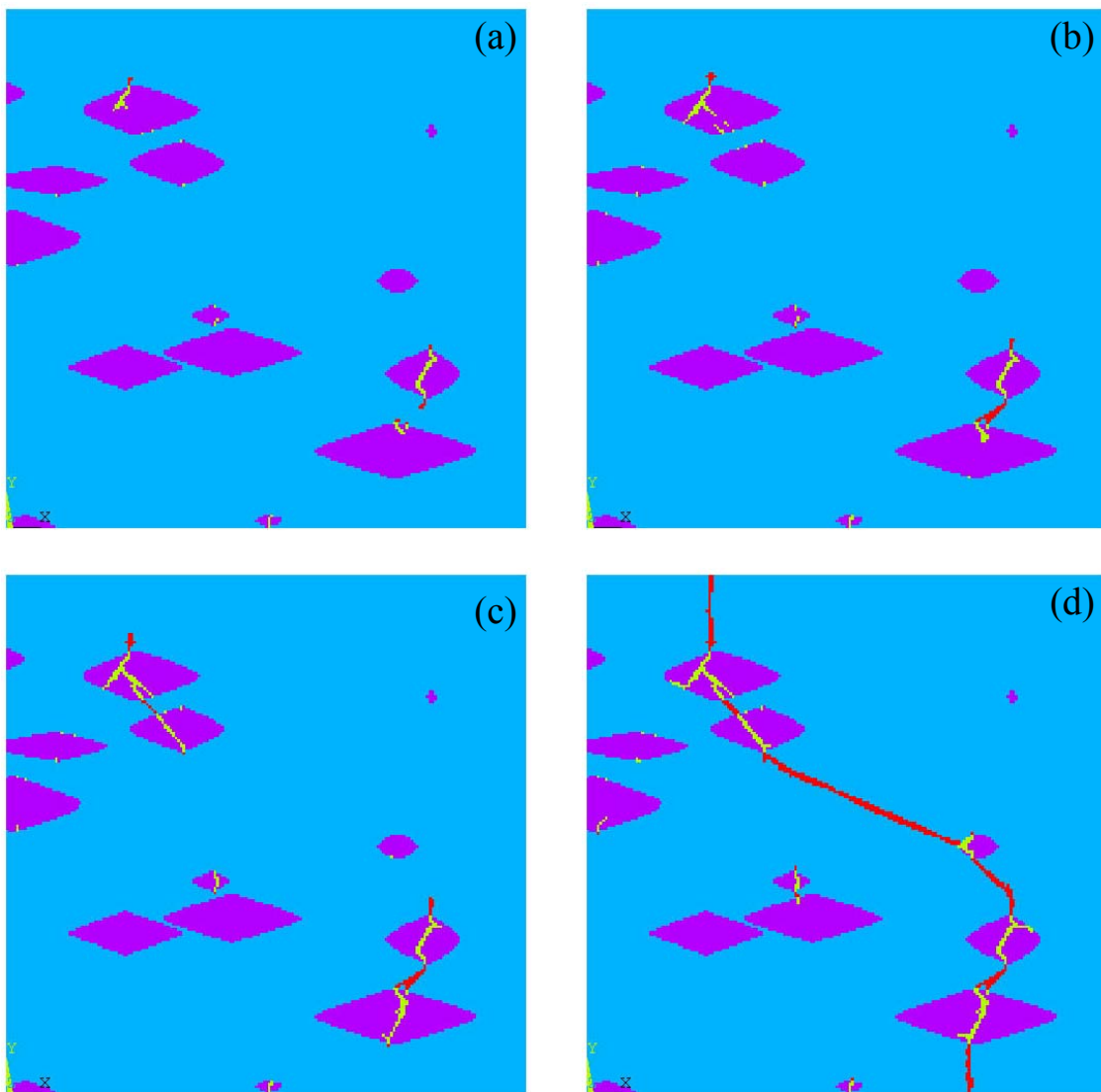


Figure 21: Crack propagation in cyclic loading after (a) 150 simulated cycles, (b) 175 simulated cycles, (c) 200 simulated cycles, and (d) 325 simulated cycles.

In summary, the higher levels of plastic strain in both tension and compression that the CGs endure cause them to accumulate damage, and ultimately fail, first. Localized regions of UFGs around the failed CG elements then undergo higher levels of plastic deformation themselves and begin to accumulate damage at higher rates. Coarse grains in close proximity to each other fail first, followed by the UFG matrix between them. Once the initial cracks have coalesced, they work their way through the remaining UFG matrix until fatigue failure occurs.

Three-Dimensional Model

Similarly to the 2D model, cracks nucleate on the edges of CGs, as shown in Figure 22(a) where 500 cycles are simulated. Fifty cycles later (Figure 22(b)), it is clear that the CGs are failing first, which substantiates the 2D results. Again, it is the higher levels of plastic strain suffered by the CGs that initiate the crack.

Three FE iterations (150 simulated cycles) later, the crack spans through both CGs, as shown in Figure 23(a), and breaches the UFG matrix 200 cycles later, as shown in Figure 23(b). The

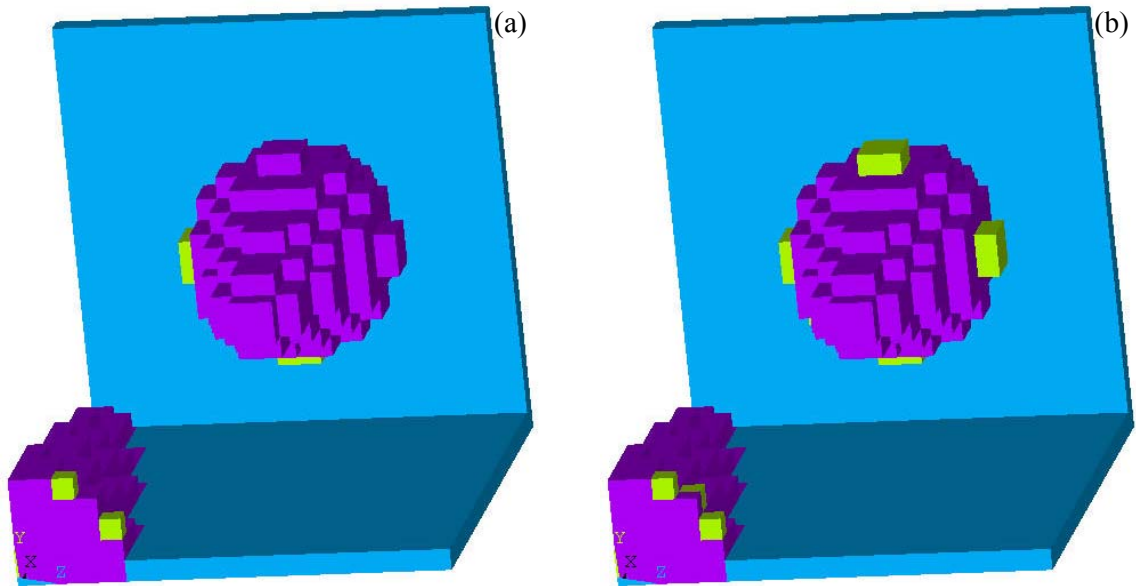


Figure 22: Crack initiation in cyclic loading after (a) 500 simulated cycles, and (b) 550 simulated cycles.

UFG matrix continues to fail, as shown in Figure 23(c) where 1050 simulated cycles are complete. Over the next four FE iterations, it grows into the large crack seen in Figure 23(d) that is about to fail the model. The three dimensional crack grows outward from the CG in a way impossible to visualize using the 2D model. Unfortunately, the level of detail possible in the 3D

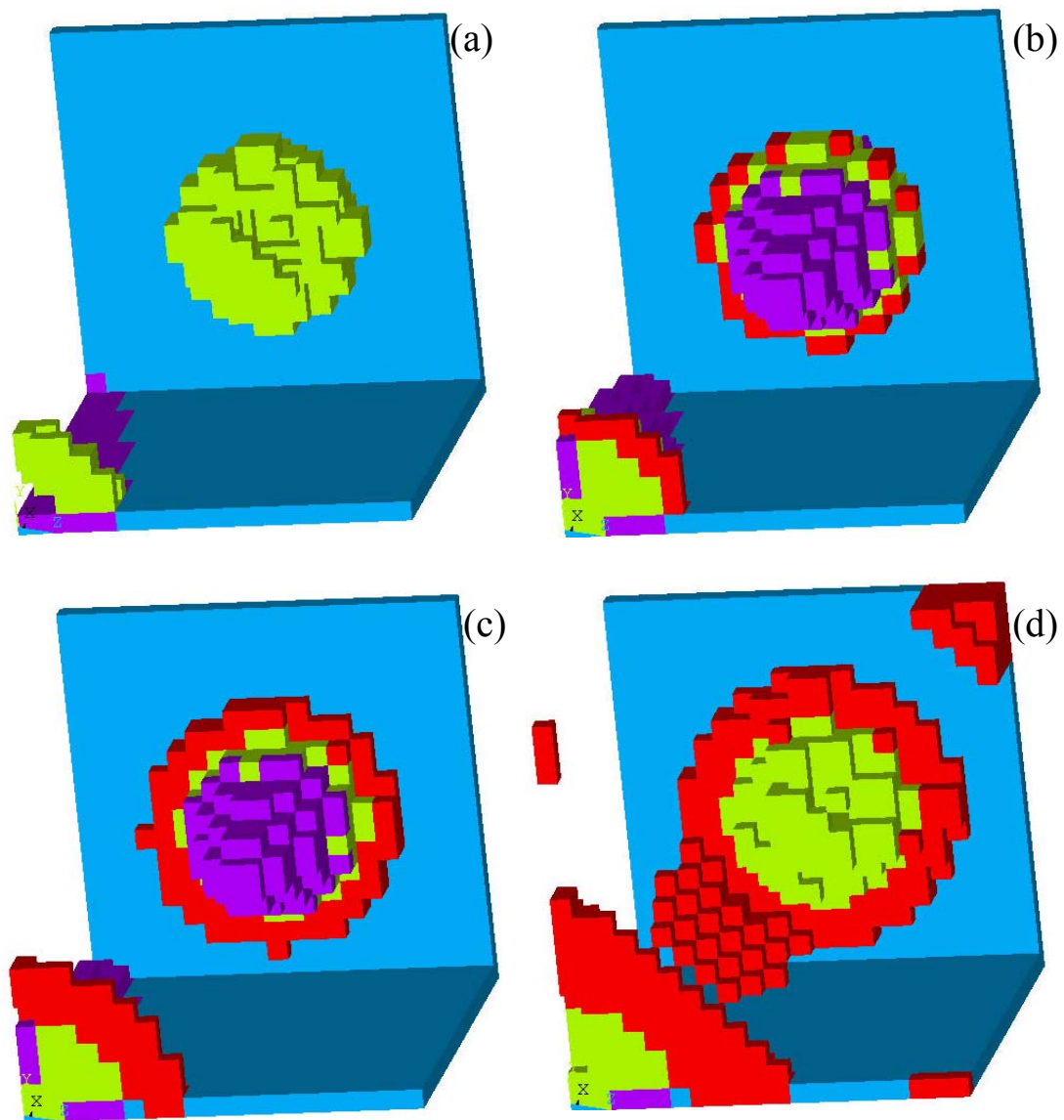


Figure 23: Crack propagation in cyclic loading after (a) 700 simulated cycles, (b) 900 simulated cycles, (c) 1050 simulated cycles, and (d) 1250 simulated cycles.

model limits the number of CGs that the crack can interact with, so essentially only the progression of the crack between two CGs is modeled in three dimensions.

The 2D and 3D fatigue models agree with and support each other in the same way the 2D and 3D tensile models do. In both, the crack nucleates within a CG at the CG-UFG interface. It progresses through first the CG regions where it nucleated and then through the surrounding UFG regions. Ultimately, cracks that started in different CGs join together and course through the rest of the material until the model fails.

Effect of Poisson Ratio Change During Plastic Deformation

The stress-strain behavior of UFG, CG and bimodal Al-5083 are described previously. There is a marked change in behavior when the transition is made from elastic deformation to plastic deformation. But the stress-strain behavior does not explicitly describe any changes in the Poisson ratio of these materials. Significant changes in failure mechanisms and crack nucleation sites could result if the Poisson ratio increases during plastic deformation.

Generally, the Poisson ratio of isotropic materials does not vary as deformation transitions from elastic to plastic. Ultra-fine grain and coarse grain materials are isotropic when investigated independently, and there are no reports of their Poisson ratios increasing during plastic deformation. Additionally, the Poisson's ratio of bimodal Al-5083 is not reported to change during plastic deformation, though no investigations have been undertaken to determine whether or not it does.

The Poisson's ratio of anisotropic materials like composite materials can change significantly after yielding occurs. Since bimodal Al-5083 fabricated in the manner described above is transversely orthotropic, there is reason for concern about the variability of its Poisson's ratio. But there is no way to know without extensive experimentation whether it changes or not, much less why and where it changes. The UFG matrix is very brittle, corresponding to a very short

region of plasticity. Since its deformation is so limited in the loading direction, there is no indication that is lateral deformation would behave differently. Additionally, the CGs behave very similarly to conventional Al-5083, whose Poisson's ratio is not reported to change during plastic deformation. But the combination of the two in a bimodal structure results in an anisotropic material, so the coupling of properties may cause a change in Poisson's ratio macroscopically. Or each of the regions could change relative to the other when coupled together.

Since there is no literature to support an assumption of a constant or varying Poisson's ratio, a sensitivity analysis is conducted. Bimodal materials loaded monotonically are much more sensitive to changes in plastic deformation, so tensile models are solved three dimensionally. Three cases of varying Poisson's ratio are solved. One where the Poisson ratio of the CGs is increased to 0.45, another where the Poisson ratio of the UFGs is increased to 0.45, and the last changes the Poisson ratios of both microstructural regions to 0.45. To accomplish this in ANSYS, the material properties are changed on a element by element basis as soon as the element's respective yield strength is reached. All properties remain the same except Poisson's ratio, which is increased after yield. Elements are eliminated in the same manner as previously described and the crack nucleation sites and path of propagation compared to the results ascertained with constant Poisson's ratio.

No deviations from the tensile results in any of the three cases were found. In all cases, the UFG matrix on the apex of a CG boundary failed first. The crack then propagated through the UFG matrix and around the CG for a time before the CG itself began to fail. So the failure mechanisms are shown to be unaffected by a change in Poisson's ratio after the transition from elastic to plastic deformation. It is interesting to note that in all of the sensitivity models the crack nucleated later in the load stepping, but propagated much faster while staying consistent with the constant Poisson ratio tensile results. The higher overall stress required for nucleation is

attributed to the boundary conditions in the local model being applied as displacements and the change in material properties. Since the boundaries are displaced a prescribed amount by the local model, the property change would allow for a state of lower stress during initial deformation. Though once the boundary displacements are large enough, the microstructure stresses and fails in the same manner.

SUMMARY AND CONCLUSIONS

Bimodal Al-5083 fails in very different ways when loaded monotonically and cyclically. Nevertheless, the difference in failure modes stem from the same microstructural characteristics. When subjected to a tensile load both the 2D and 3D FE models agree that the crack nucleates in ultra-fine grain regions that are adjacent to coarse grains. In both models the crack propagates away from or around the CG before directly afflicting it. Crack propagation then accelerates as elements in the model fail until the crack spans the model, running through both microstructural regions. The 2D HIPped tensile model does not agree fully with these conclusions, but the differences are attributed to the uncharacteristically high levels of ductility reported for the HIPped material. Therefore, they do not invalidate the findings reported in the remaining tensile models.

Conversely, when subjected to cyclic loads, the crack nucleates at the outer edges of the CGs in both the 2D and 3D models. In the 2D model it traverses the affected CGs before expanding outward into the UFG matrix, whereas in the 3D model only one CG is damaged initially and the crack propagates into the UFGs from there before the other CG is affected. Model size and resolution limit the number of CGs in the 3D model, limiting the correlation between the two models, but it is reasonable to expect the same behavior in the 3D model if it were possible to model more CGs without exceeding memory limitations. Once the crack connects the severely damaged CGs, the crack expands through the UFG matrix until the model fails entirely.

These different locations of crack nucleation and modes of propagation, though both due to the CG-UFG matrix interface, are symptoms of load. When loaded in tension, the mismatch in properties, namely the ductility, across the CG boundary results in the UFG region being stressed beyond its ultimate strength sooner than the CG regions. At a given value of stress, the CG

elements deform more, so the UFG elements adjacent to them will be stressed even more, leading to their earlier failure.

On the other hand, when subjected to cyclic loads, the plastic strain amplitude is held constant. Macroscopically, the amount of plastic deformation is more dependent upon the UFGs than the CGs. Since the yield strength of the UFG region is higher, they will continue to deform elastically as the CGs reach their yield strength and start to deform plastically. For the same macroscopic strain amplitude, the CG elements experience significantly more plastic deformation in both the tensile and compressive cycles. Also, since the damage models are nearly equivalent in both regions, the fatigue lives of the CGs are severely reduced. Consequentially, the CG elements accumulate more damage sooner and faster than the UFG elements.

These findings stress the importance of determining the proper CG content when using a bimodal material. Of course the macroscopic properties of bimodal materials will primarily affect the choice of CG content as the required strengths and ductility will drive the choice. In tensile loading, it is safe to assume that the UFGs will always fail first, no matter the CG content, so for monotonic loading the strength required drives choice of bimodal material. But in cyclic loading, the CGs themselves are the catalyst for crack nucleation. Macroscopically, the fatigue behavior of all levels of CG content are about the same, but since the plastic deformation occurring locally severely limits the fatigue lives of CGs when interspersed between UFGs, there is a reasonable concern that bimodal Al-5083 will not perform as well as UFG or CG Al-5083 alone. Here, only a very low plastic strain amplitude (0.1%) is tested, and at plastic strain amplitudes this low the difference between the plastic strain amplitudes felt by the CGs and UFGs is high. At higher plastic strain amplitudes, this difference will not be so pronounced and these results may not be applicable. But at the upper limit of the low cycle fatigue regime, the addition of CGs can cause the inevitable fatigue micro-cracks to nucleate far earlier than they

would otherwise. Even if the material does not fail any sooner macroscopically, the internal microstructural damage is not desirable in any loading scenario.

In conclusion, the path of propagation in monotonic loading scenarios, though interesting to know, does not significantly affect the choice of CG content in bimodal Al-5083. But in cyclic loading scenarios they are the medium in which fatigue micro-cracks nucleate, so their addition will cause undesirable, even premature, levels of damage to accumulate in the microstructure. Though the macroscopic fatigue lives of UFG, CG and bimodal Al-5083 do not vary a lot, any amount of damage very early in the fatigue lives of materials will make them very susceptible to any loading variations and cause untimely failure. This is of particular concern at low plastic strain amplitudes in the low cycle fatigue regime.

REFERENCES

- [1] Hall, E. O., 1951, "The Deformation and Ageing of Mild Steel: III Discussion of Results," Proc. Physical Society B, 64(9), pp. 747-753.
- [2] Petch, N. J., 1953, "The Cleavage Strength of Polycrystals," J. Iron and Steel Institute, 174, pp. 25-28.
- [3] Han, B. Q., Lee, Z., Nutt, S. R., Lavernia, E. J., and Mohamed, F. A., 2003, "Mechanical Properties of An Ultrafine-Grained Al-7.5 Pct Mg Alloy," Metallurgical and Materials Trans. A, 34A, pp. 603-613.
- [4] Han, B. Q., Mohamed, F. A., Bampton, C. C., and Lavernia, E. J., 2005, "Improvement of Toughness and Ductility of a Cryomilled Al-Mg Alloy via Microstructural Modification," Metallurgical and Materials Trans. A, 36A, pp. 2081-2091.
- [5] Ahn, B., Newbery, A. P., Lavernia, E. J., and Nutt, S. R., 2007, "Effect of Degassing Temperature on the Microstructure of a Nanocrystalline Al-Mg Alloy," Materials Sci. Eng., 463, pp. 61-66.
- [6] Fan, G. J., Choo, H., Liaw, P. K., and Lavernia, E. J., 2006, "Plastic Deformation and Fracture of Ultrafine-grained Al-Mg Alloys with a Bimodal Grain Size Distribution," Acta Materialia, 54, pp. 1759-1766.
- [7] Youssef, K. M., Scattergood, R. O., Murty, K. L., and Koch, C. C., 2006, "Nanocrystalline Al-Mg alloy with Ultrahigh Strength and Good Ductility," Scripta Materialia, 54, pp. 251-256.
- [8] Newbery, A. P., Ahn, B., Topping, T. D., Pao, P. S., Nutt, S. R., and Lavernia, E. J., 2008, "Large UFG Al Alloy Plates from Cryomilling," J. Materials Processing Tech., 203, pp. 37-45.

- [9] Lavernia, E. J., Han, B. Q., and Schoenung, J. M., 2008, "Cryomilled Nanostructured Materials: Processing and Properties," *Materials Sci. Eng.*, 493, pp. 207-14.
- [10] Han, B. Q., Ye, J., Tang, F., Schoenung, J., and Lavernia, E. J., 2007, "Processing and Behavior of Nanostructured Metallic Alloys and Composites by Cryomilling," *J. Material Sci.*, 42, pp. 1660-672.
- [11] Witkin, D. B., and Lavernia, E. J., 2006, "Synthesis and Mechanical Behavior of Nanostructured Materials via Cryomilling," *Progress in Materials Sci.*, 51, pp. 1-60.
- [12] Lee, Z., Witkin, D. B., Radmilovic, V., Lavernia, E. J., and Nutt, S. R., 2005, "Bimodal Microstructure and Deformation of Cryomilled Bulk Nanocrystalline Al-7.5Mg Alloy," *Materials Sci. Eng. A*, 410, pp. 462-467.
- [13] Newbery, A. P., Ahn, B., Hayes, R. W., Pao, P. S., Nutt, S. R., and Lavernia, E. J., 2008, "Consolidation and Forging Methods for a Cryomilled Al Alloy," *Metallurgical and Materials Trans. A*, 39A, pp. 2193-2205.
- [14] Walley, J. L., Lavernia, E. J., and Gibeling, J. C., 2009, "Low-Cycle Fatigue of Ultra-Fine-Grained Cryomilled 5083 Aluminum Alloy," *Metallurgical and Materials Trans. A*, 40A, pp. 2622-2630.
- [15] Ye, J., Ajdelsztajn, L., and Schoenung, J. M., 2006, "Bulk Nanocrystalline Aluminum 5083 Alloy Fabricated by a Novel Technique: Cryomilling and Spark Plasma Sintering," *Metallurgical and Materials Trans. A*, 37A, pp. 2569-2579.
- [16] Ahn, B., Lavernia, E. J., and Nutt, S. R., 2008, "Dynamic observations of deformation in an ultrafine-grained Al-Mg alloy with bimodal grain structure," *J. Material Sci.*, 43, pp. 7403-7408.
- [17] Ahn, B., and Nutt, S. R., 2009, "Strain Mapping of Al-Mg Alloy with Multi-scale Grain Structure Using Digital Image Correlation Method," *Experimental Mechanics*, 50(1), pp. 117-123.

- [18] Han, B. Q., Lee, Z., Witkin, D., Nutt, S.R., and Lavernia, E. J., 2005, "Deformation Behavior of Bimodal Nanostructured 5083 Al Alloys," *Metallurgical and Materials Trans. A*, 36A, 957-965.
- [19] Hayes, R. W., Witkin, D., Zhou, F., and Lavernia, E. J., 2004, "Deformation and Activation Volumes of Cryomilled Ultrafine-grained Aluminum," *Acta Materialia*, 52, pp. 4259-4271.
- [20] Tellkamp, V. L., Melmed, A., and Lavernia, E. J., 2001, "Mechanical Behavior and Microstructure of a Thermally Stable Bulk Nanostructured Al Alloy," *Metallurgical and Materials Trans. A*, 32A, pp. 2335-2343.
- [21] Witkin, D., Lee, Z., Rodriguez, R., Nutt, S. R., and Lavernia, E. J., 2003, "Al-Mg Alloy Engineered with Bimodal Grain Size for High Strength and Increased Ductility," *Scripta Materialia*, 49, pp. 297-302.
- [22] Witkin, D., Han, B. Q., and Lavernia, E. J., 2006, "Room-Temperature Mechanical Behavior of Cryomilled Al Alloys," *Metallurgical and Materials Trans. A*, 37A, pp. 185-194.
- [23] Pao, P. S., Jones, H. N., Cheng, S. F., and Feng, C. R., 2005, "Fatigue Crack Propagation in Ultrafine Grained Al-Mg Alloy," *Intl. J. Fatigue*, 27, pp. 1164-1169.
- [24] Joshi, S. P., Ramesh, K. T., Han, B. Q., and Lavernia, E. J., 2006, "Modeling the Constitutive Response of Bimodal Metals," *Metallurgical and Materials Trans. A*, 37A, pp. 2397-2404.
- [25] Fan, G. J., Wang, G. Y., Choo, H., Liaw, P. K., Park, Y. S., Han, B. Q., and Lavernia, E. J., 2005, "Deformation Behavior of an Ultrafine-grained Al-Mg Alloy at different Strain Rates," *Scripta Materialia*, 52, pp. 929-933.
- [26] Padilla, H. A., and Boyce, B. L., 2010, "A Review of Fatigue Behavior in Nanocrystalline Metals," *Experimental Mechanics*, 50, pp. 5-23.

- [27] Hoppel, H. W., Kautz, M., Xu, C., Murashkin, M., Langdon, T. G., Valiev, R. Z., and Mughrabi, H., 2006, "An Overview: Fatigue Behaviour of Ultrafine-grained Metals and Alloys," *Intl. J. Fatigue*, 28, pp. 1001-1010.
- [28] Mughrabi, H., Hoppel, H. W., and Kautz, M., 2004, "Fatigue and Microstructure of Ultrafine-grained Metals Produced by Severe Plastic Deformation," *Scripta Materialia*, 51, pp. 807-812.
- [29] Patlan, V., Vinogradov, A., Higashi, K., and Kitagawa, K., 2001, "Overview of Fatigue Properties of Fine Grain 5056 Al-Mg Alloy Processed by Equal-channel Angular Pressing," *Materials Sci. Eng. A*, 300, pp. 171-82.
- [30] Avtokratova, E. V., Kaibyshev, R. O., and Sitdikov, O. Sh., 2008, "Fatigue of a Fine-Grained High-Strength Al-6Mg-Sc Alloy Produced by Equal-Channel Angular Pressing," *The Physics of Metals and Metallography*, 105(5), pp. 500-508.
- [31] Ding, H. Z., Mughrabi, H., and Hoppel, H. W., 2002, "A Low-cycle Fatigue Life Prediction Model of Ultrafine-grained Metals," *Fatigue and Fracture of Eng. Materials and Structures*, 25, pp. 975-984.
- [32] Voce, E., 1948, "The Relationship Between Stress and Strain for Homogeneous Deformation," *J. Institute of Metals*, 74, pp. 537-562.
- [33] Rice, J. R., 1968, "A Path Independent Integral and the Approximate Analysis of Strain Concentration by Notches and Cracks," *J. Appl. Mechanics*, 35, pp. 379-386.
- [34] Ladani, L. J., and Dasgupta, A., 2008, "Damage Initiation and Propagation in Voided Joints: Modeling and Experiment," *ASME J. Electronic Packaging*, 130(1), Paper No. 011008-1-11.
- [35] Ladani, L. J., 2008, "Reliability Estimation for Large-area Solder Joints Using Explicit Modeling of Damage," *IEEE Trans. on Device and Materials Reliability*, 8(2), pp. 375-386.

- [36] Ladani, L. J., 2010, "Successive Softening and Cyclic Damage in Visco-plastic Material," ASME J. Electronic Packaging, Accepted, May 2010.
- [37] Gyllenskog, J., and Ladani, L. J., 2010, "Continuum Damage Modeling of Fatigue Crack Initiation and Propagation in T-38 Aileron Lever and Comparison with Field Data," To be submitted, June 2010.
- [38] Gyllenskog, J., 2010, "Fatigue Life Analysis of T-38 Aileron Lever Using a Continuum Damage Approach," M.S. thesis, Mechanical and Aerospace Engineering Department, Utah State University.

APPENDICES

Appendix A: Low resolution Finite element results

This appendix contains many microstructural FE figures, all of which are color coded in the same manner. Ultra-fine grain regions are shown in green and coarse grain regions in purple. Red failure paths indicate eliminated UFG elements, and blue failure paths denote eliminated CG elements. This color scheme applies to both the 2D and 3D models. Also, it is difficult to illustrate the path of crack propagation in the 3D models using 2D figures. So, for the sake of visual clarity, the failed elements and the CG elements are shown framed by only the outer edge of the model which is comprised of almost entirely UFG elements. Though only the outer edge UFG elements are shown, all the empty space in the figures is occupied by live UFG elements. Additionally, for the 2D models, a grid size of 100×100 is used, making the size of each element in the models $100 \times 100 \text{ nm}^2$. The 3D models have grid sizes of $15 \times 15 \times 15$, or individual element sizes of $667 \times 667 \times 667 \text{ nm}^3$.

Tensile Failure of CIPped Al-5083

Two-Dimensional Model

To model the tensile failure in two dimensions loads are applied to the global model starting at the macroscopic yield strength (560 MPa) and ending at the macroscopic ultimate strength (690 MPa) of bimodal Al-5083 comprised of ten percent CGs. Fifteen steps are solved, each accounting for 14.75 MPa of sequential loading.

The crack initiates after the third load step when the central gage stress in the global model is 584 MPa. It is shown in Figure 24(a) as a single failed UFG element on the boundary of a CG. Stresses throughout the local model are shown in Figure 24(b), and it is immediately apparent that the high stress regions are localized to the UFG matrix near the CG boundaries. Many of these regions are very near the UTS (690 MPa) of the UFG elements, indicating that failure could be imminent in all of these high stress areas. Also, the stresses in the CGs are well below the CG

ultimate tensile strength of 390 MPa. The lower stress in the CGs is directly attributed to their higher level of ductility. In other words, given equivalent strains in the UFG and CG elements, the corresponding stress level in the UFGs will be much closer to their ultimate strength than the stress level in the CGs.

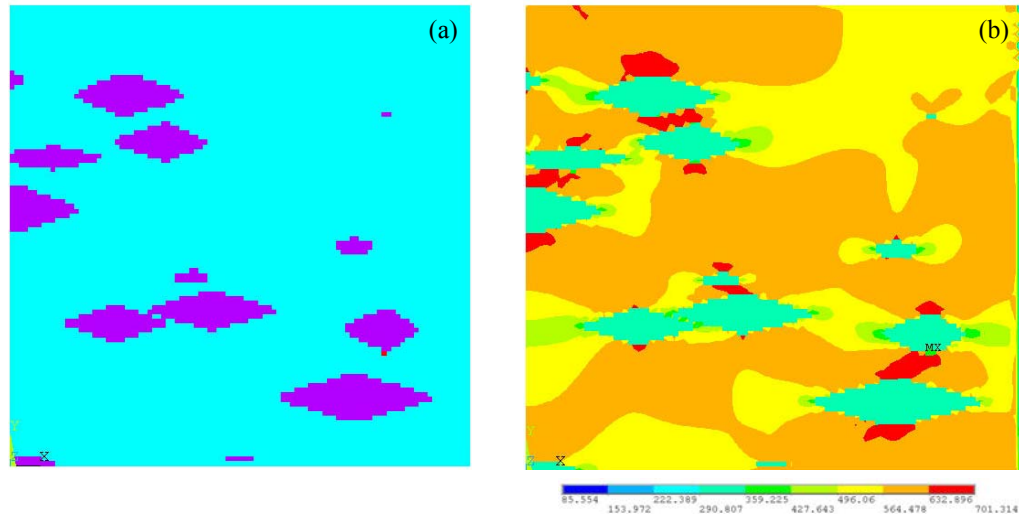


Figure 24: Crack initiation in tension corresponding to a gage section stress of 584 MPa (load step 3) in the global model; (a) eliminated elements in the microstructure, (b) stress fields in the UFG matrix and CG regions.

One load step later, the crack grows in the UFG matrix, extending towards the nearest neighboring CG, as shown in Figure 25(a). It also breaches the boundary into the CG it started next to. Referring to the stress field shown in Figure 25(b), the high stress regions in the UFG matrix have grown, and the stress levels throughout the CGs have increased to very near their ultimate strength.

The crack after the fifth and seventh load steps is shown in Figure 26(a) and Figure 26(b), respectively. After the fifth load step the connection between the CGs is complete, and both ends of the crack have slashed into the CGs. Two steps later the crack extends through both CGs and the UFG matrix on both sides. It stops at the boundary of the next nearest CG, and a symmetry

edge of the model. Also, note that more cracks are initiating on CG boundaries in a different region of the model.

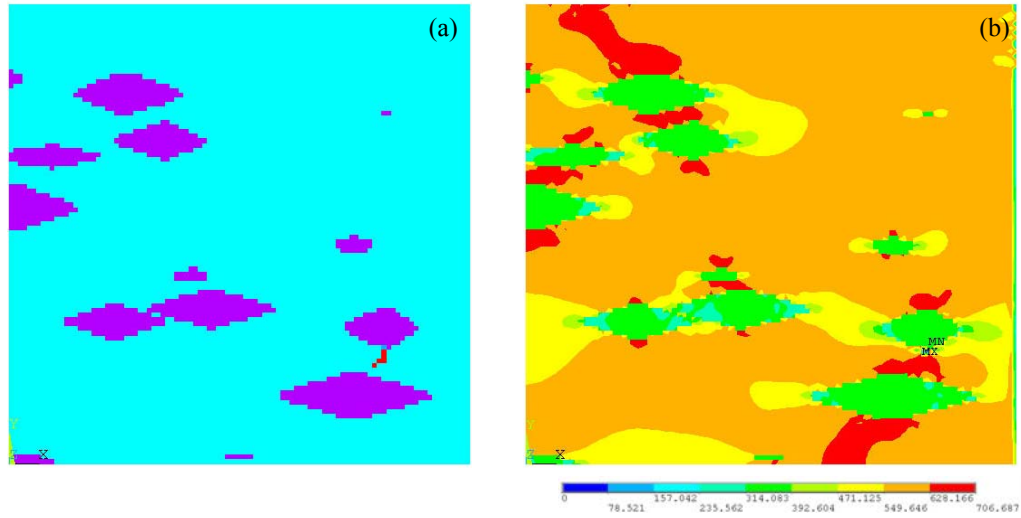


Figure 25: Crack propagation in tension corresponding to a gage section stress of 595 MPa (load step 4) in the global model; (a) eliminated elements in the microstructure, (b) stress fields in the UFG matrix and CG regions.

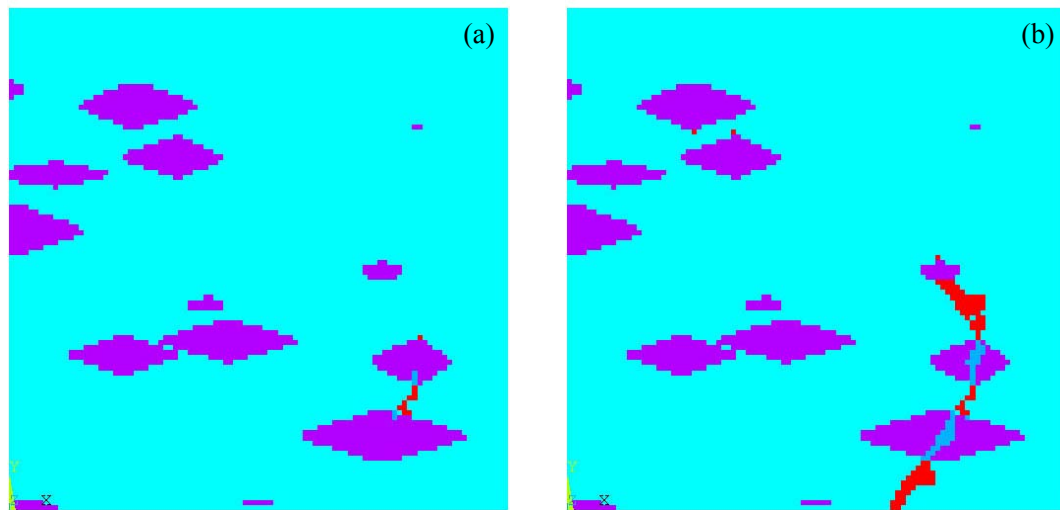


Figure 26: Crack propagation in tension corresponding to gage section stresses of (a) 607 MPa (load step 5) and (b) 619 MPa (load step 7) in the global model.

Three steps later, the new cracks have spanned the distance between their respective CGs as well as entering the CGs (Figure 27(a)). Also, the main crack now extends across most of the model, recently slicing through the small CG that had impeded its progression three steps earlier. It is on the verge of extending throughout the entire model, as is evident by the newest crack initiating on the edge of the partial CG. The model fails two steps later when the stress in the global model's gage section is 690 MPa, the global ultimate strength, as shown in Figure 27(b).

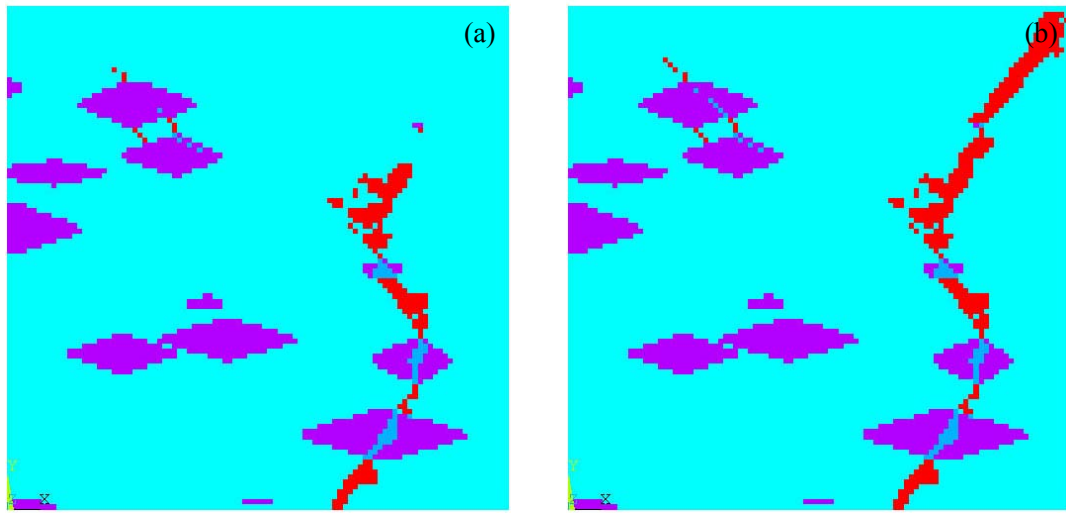


Figure 27: Crack propagation in tension corresponding to gage section stresses of (a) 666 MPa (load step 10) and (b) 690 MPa (load step 12) in the global model.

In summary, the crack starts at a boundary between the UFG matrix and a CG. This occurs because the brittle ultra-fine grains cannot withstand deformation as well as the ductile coarse grains. Once the crack is started, the surrounding UFGs fail very soon, and the stress concentrations at the crack tip that is pressed up against a CG cause said CG to fail. This process repeats itself as the crack propagates through the UFG matrix towards surrounding CGs, swallowing them up as soon as contact is made.

Three-Dimensional Model

The three dimensional successive initiation solution is determined by increasing the load for 30 steps from the global yield strength (560 MPa) to the global ultimate strength (690 MPa) of bimodal Al-5083 comprised of ten percent CGs. This results in a step size of 4.3 MPa applied sequentially.

Similarly to the 2D model, the crack initiates in the UFG matrix at the boundary of a CG, as shown in Figure 28(a). A cross sectional slice of the model showing the stress distribution in the UFG matrix surrounding the initial crack is shown in Figure 28(b). Coarse grain regions are omitted to provide a clear picture of the stress in the UFG matrix. The stresses in the CGs are far lower than their ultimate strength and do not largely impact the stresses at the location of crack initiation, and therefore are not shown here. The stresses in the UFGs increase with their proximity to the CG, peaking at the boundary, particularly where the failure starts. As stated in previously in the 2D model, UFG stresses at the interface are significantly higher because the

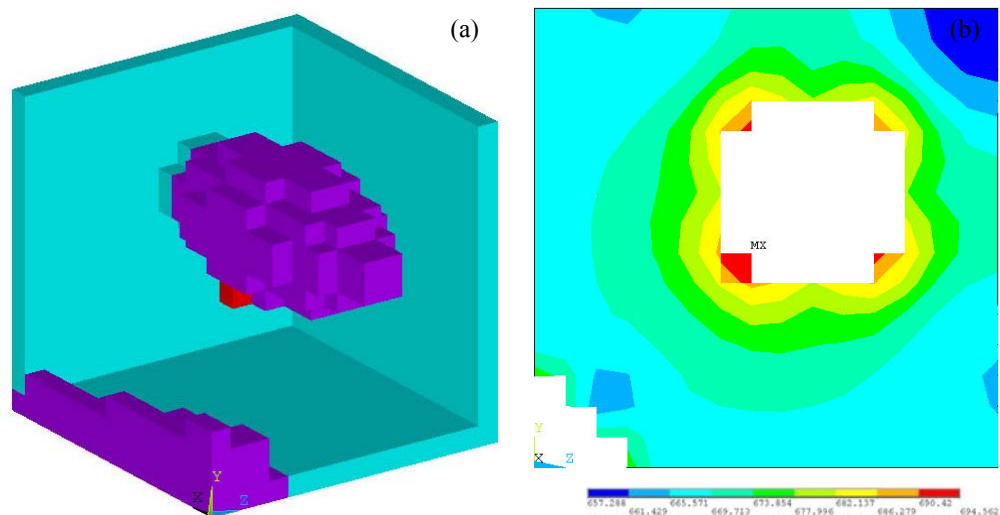


Figure 28: Crack initiation in tension corresponding to a gage section stress of 599 MPa (load step 10) in the global model; (a) eliminated elements in the microstructure, (b) stress fields in a UFG matrix cross section taken at the x-location of the initial crack.

interactions at the boundary cause the UFG region to displace more than it would in the absence of the CG. This increase in strain corresponds to a higher level of stress, and ultimately causes the UFG material to fail at the boundary.

Three load steps later, more UFG elements surrounding the site of initiation have failed, as have elements surrounding the other CG at the symmetry corner of the model (Figure 29(a)). Ultra-fine grain elements between the two initiation sites are also failing, indicating the initial path of propagation. This is further illustrated an additional step later by Figure 29(b) where the connection between the CGs is almost complete. Unseen in these figures is the failure of elements in the center CG, which is also occurring.

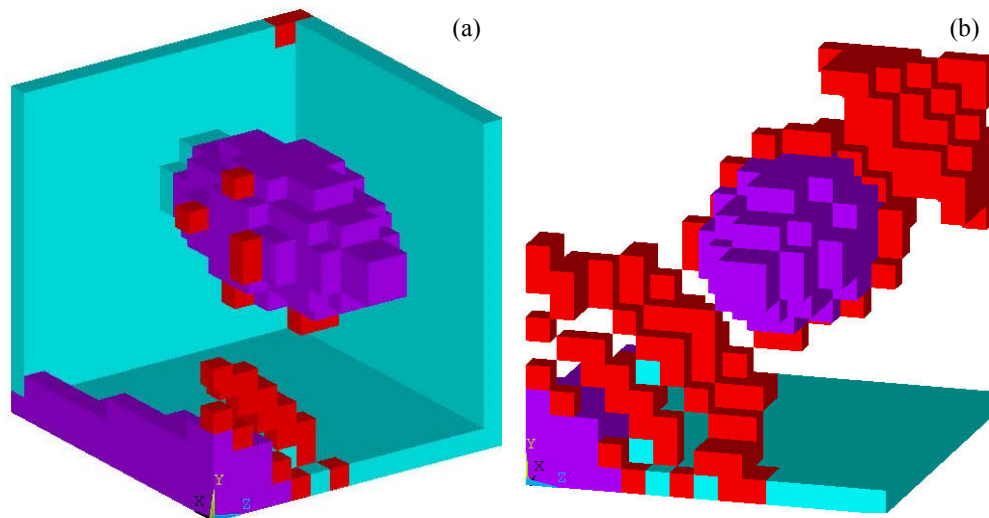


Figure 29: Crack propagation in tension corresponding to gage section stresses of (a) 612 MPa (load step 13) and (b) 616 MPa (load step 14) in the global model.

One step later, the crack spans the length of the model as shown in Figure 30(a), and the affliction to the CGs is finally visible, particularly in the CG on the symmetry boundaries. At this point, the model is as good as failed, but through the course of 15 more steps, small amounts of

additional failure do occur, as shown in Figure 30(b), before the model is unable to solve anything further.

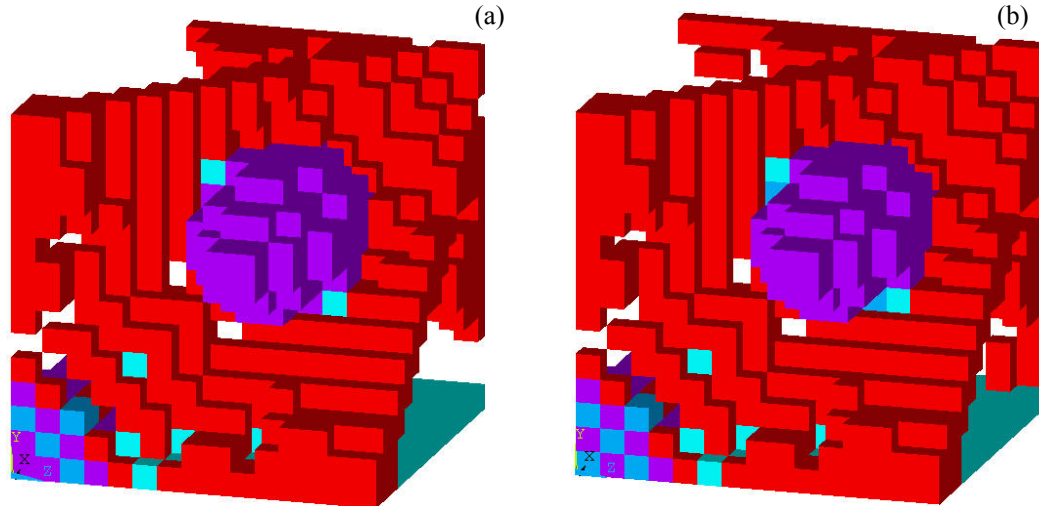


Figure 30: Crack propagation in tension corresponding to gage section stresses of (a) 620 MPa (load step 15) and (b) 690 MPa (load step 30) in the global model.

The two and three dimensional models agree very well in that crack initiation occurs in the UFG matrix on the boundary of a CG. Furthermore, there are multiple points of nucleation after initiation. In both cases the cracks initially move through the UFG matrix before afflicting the CG regions, and once the CGs are affected, both models show that they fail very quickly thereafter. Failure is attributed to the brittleness of the UFGs. Given the same displacements, they will reach their ultimate strength much sooner and fail. The crack that results from their failure creates zones of concentrated stress that cause a cascade effect on both the surrounding UFGs and CGs.

Tensile Failure of HIPped Al-5083

In both the 2D and 3D models, the loads are applied to the global model starting at the macroscopic yield strength (380 MPa) and ending at the macroscopic ultimate strength (450

MPa) of bimodal Al-5083 comprised of ten percent CGs. Ten steps are solved, each accounting for 7 MPa of sequential loading.

Two-Dimensional Model

The crack nucleates on a CG boundary simultaneously in both the CG and UFG regions after four load steps (401 MPa of global stress), as shown in Figure 31(a). Stresses in the model are shown in Figure 31(b) and are concentrated in the UFG regions between the CGs, particularly in the failing region.

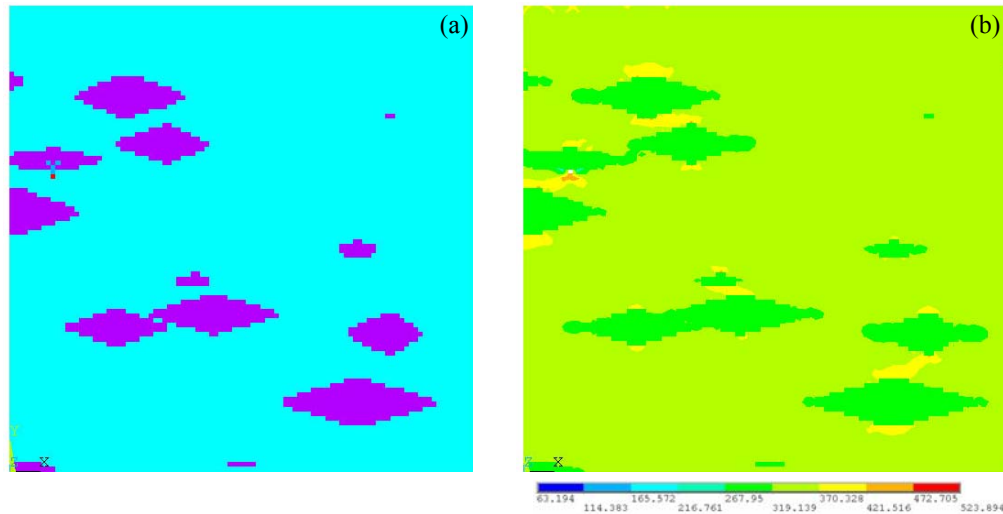


Figure 31: Crack propagation in tension corresponding to a gage section stress of 401 MPa (load step 4) in the global model; (a) eliminated elements in the microstructure, (b) stress fields in the UFG matrix and CG regions.

Figure 32(a) and Figure 32(b) show the next two loads steps (five and six) that correspond to stresses in the global model of 408 and 415 MPa, respectively. At first the crack expands through the CG where it was nucleated and spans the UFG matrix into the adjacent CG as well. Then, as loading continues, it grows significantly, affecting two additional CG and the UFG matrix in between.

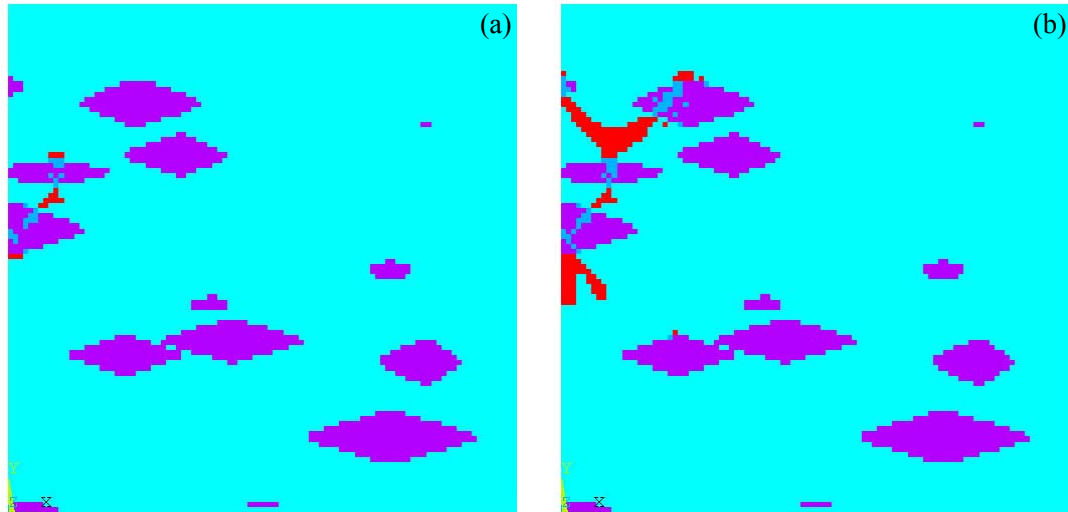


Figure 32: Crack propagation in tension corresponding to gage section stresses of (a) 408 MPa (load step 5) and (b) 415 MPa (load step 6) in the global model.

It only takes two more load steps before the model fails entirely. The size of the crack when the stress in the global model has reached 422 MPa is shown in Figure 33(a) and Figure 33(b) shows the crack after the model fails at a global stress of 429 MPa.

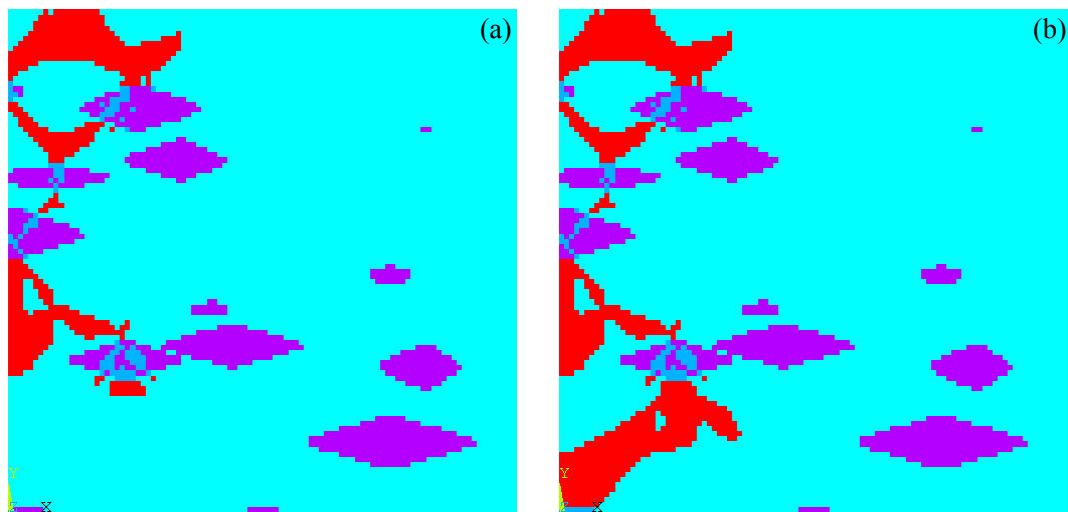


Figure 33: Crack propagation in tension corresponding to gage section stresses of (a) 422 MPa (load step 7) and (b) 429 MPa (load step 8) in the global model.

Note that failure of the HIPped material model occurred far faster than in the CIPped material model, and that both the UFG and CG regions failed simultaneously in this HIPped model. The CIPped models demonstrated that the UFG regions fail first, and these results directly conflict with that. This is likely due to the uncharacteristic material properties reported for the HIPped material, and not a typical result

Three-Dimensional Model

The behavior of this HIPped material is more closely comparable to the CIPped material in the 3D model. Here, the crack initiates in the UFG matrix on the edges of both CGs after the first load step (global stress of 380 MPa), as shown in Figure 34(a). The main difference is the earlier nucleation time which occurs at the global yield strength for the HIPped material, but well above that for the CIPped material. Figure 34(b) illustrates the stress field in the UFG matrix around the CGs and confirms that the failing elements are those exhibiting the highest values of stress.

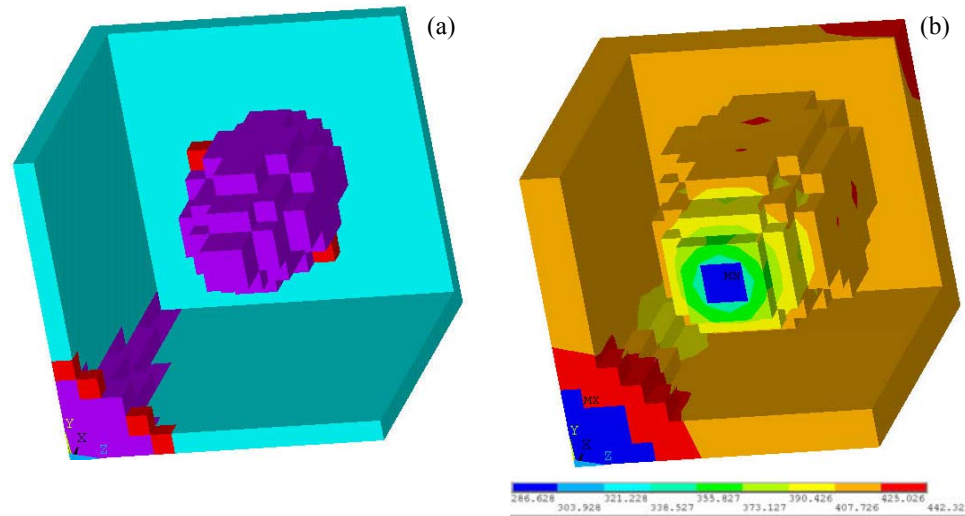


Figure 34: Crack initiation in tension corresponding to a gage section stress of 380 MPa (load step 1) in the global model; (a) eliminated elements in the microstructure, (b) stress fields in the UFG matrix on the edges of the CGs.

A single load step later (387 MPa of global stress), the crack is propagating quickly outward, away from the CG boundary nucleation sites (Figure 35(a)). When one more load step is complete (394 MPa of global stress), the CG on the symmetry axes is failing, and the central CG is virtually surrounded by cracks in the UFG matrix, though it is not yet cracking itself (Figure 35(b)).

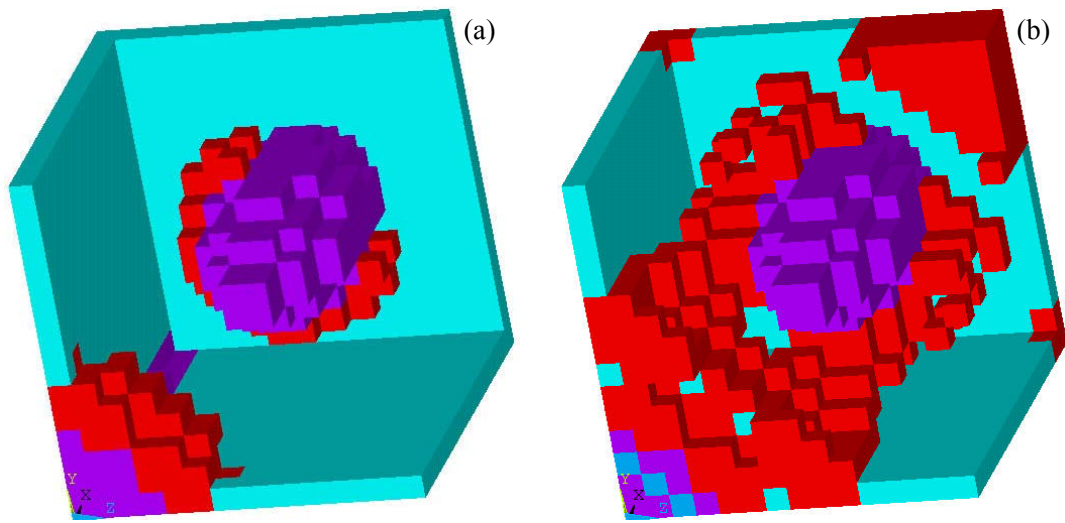


Figure 35: Crack propagation in tension corresponding to gage section stresses of (a) 387 MPa (load step 2) and (b) 394 MPa (load step 3) in the global model.

Failure of the HIPped 3D model occurs only a single load step later, when the global stress is 401 MPa. Figure 36(a) shows this failure relative to the CGs and Figure 36(b) shows the failure within the CGs. The accelerated rate at which this model failed conflicts with the failure rate of the CIPped models, but is attributed to the lower yield strengths and uncharacteristic high elongations at failure that the HIPped materials are reported to have relative to the reported properties of the CIPped material. It is reassuring to note that the nucleation site and path of crack propagations both agreed with the other tensile models fully.

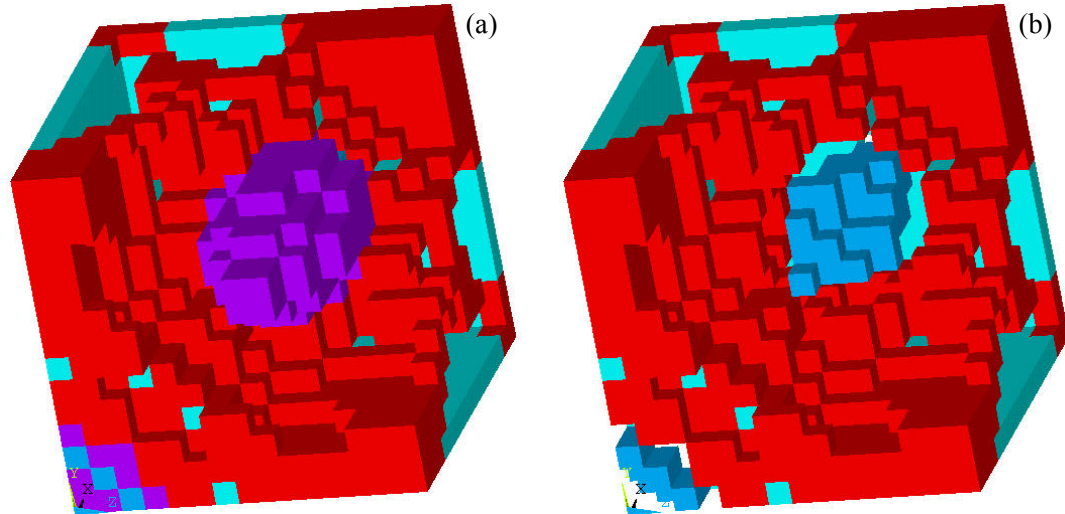


Figure 36: Crack propagation in tension corresponding to gage section stresses of 401 MPa (load step 4): (a) crack relative to CGs and (b) failed regions within the CGs.

Cyclic Failure of HIPped Al-5083

The successive initiation solution for a cyclic loading scenario is determined using tensile and compressive displacements obtained from the global model given a plastic strain amplitude of 0.01 and applying them to the local model. Each FE iteration simulates 200 cycles of fatigue loading with damage accumulated in an element by element basis. All elements start with an initial damage of zero, and are considered to have failed when their damage value equals or exceeds unity.

Two-Dimensional Model

Initially, fatigue micro-cracks nucleate within the CGs where they intersect with the UFG matrix, as shown in Figure 37(a). Part of virtually every CG is damaged after the first 200 simulated cycles, with the locations of maximum damage shown in Figure 37(b). Most of the model remains undamaged through the first iteration, but isolated areas corresponding with the CG boundaries are seriously afflicted by damage. This observation is in direct contradiction to the monotonic results previously presented, but is not unexpected. When subjected to equivalent

global plastic strain amplitudes, the CG regions experience higher levels of plastic deformation than the UFG regions. Since the UFG and CG damage models do not differ in a large way, the higher plastic strain amplitude the CGs endure reduce their fatigue lives significantly therefore increasing the damage they accumulate to the point of failure.

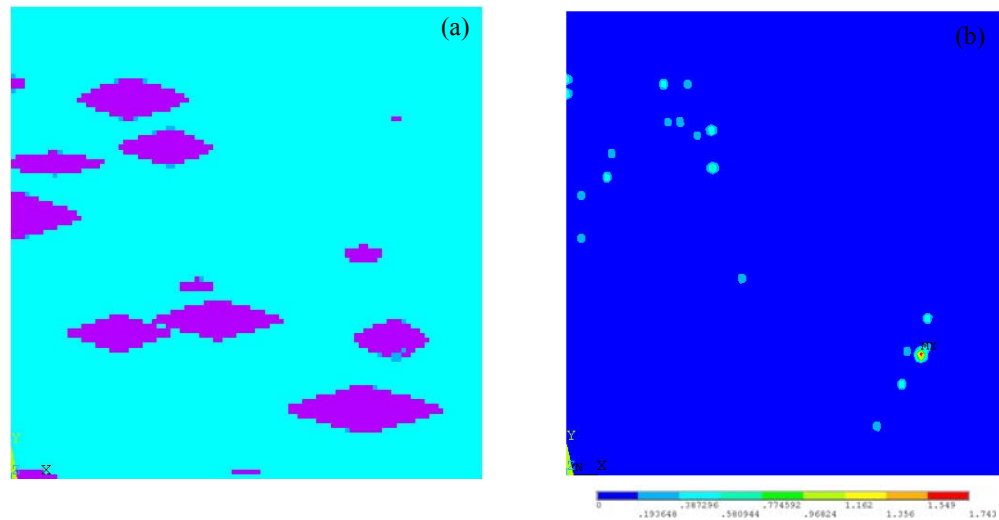


Figure 37: Crack initiation in cyclic loading after 200 simulated cycles; (a) eliminated elements in the microstructure, (b) sites of maximum accumulated damage.

After 200 additional simulated cycles, the multiple nucleated micro-cracks begin to coalesce into fracture systems (Figure 38(a)). Every CG afflicted by high levels of damage has cracked through. Additionally, cracks have formed between affected CGs in close proximity to each other, quickly spanning the UFG matrix in between. These high levels of damage are further illustrated in Figure 38(b), where bands of high damaged areas correspond to the fracture pattern in Figure 38(a).

When the simulated cycles reach 1000, the larger gaps between damaged CGs are spanned, as shown in Figure 39(a). Now the crack spans most of the model and the crack grows at a much slower rate. This is shown in Figure 39(b) where 5000 more simulated cycles (25 FE iterations)

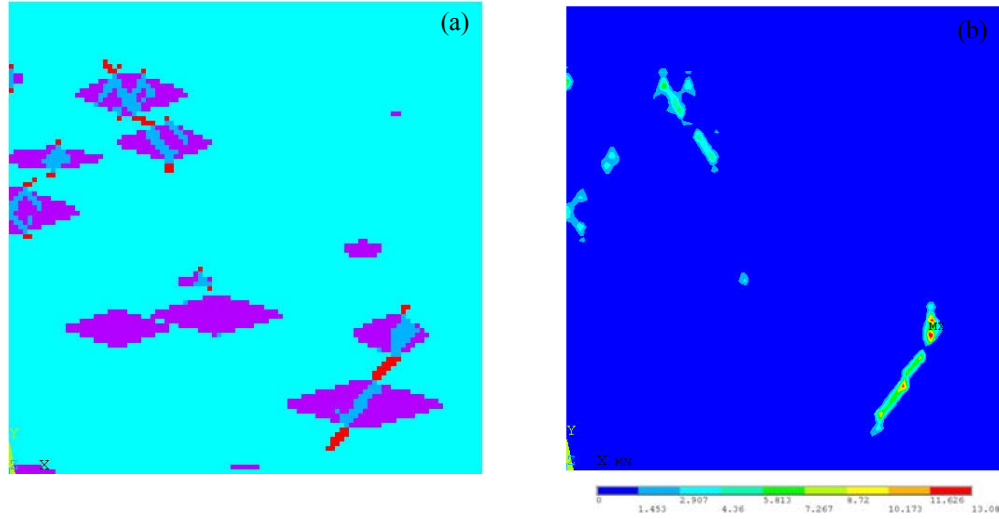


Figure 38: Crack propagation in cyclic loading after 400 simulated cycles; (a) eliminated elements in the microstructure, (b) sites of maximum accumulated damage.

are accumulated. One last connection is made between CGs, and two other long connections slowly extend towards one another, indicating imminent failure of the model.

In summary, the higher levels of plastic strain in both tension and compression that the CGs endure cause them to accumulate damage, and ultimately fail, first. Localized regions of UFGs

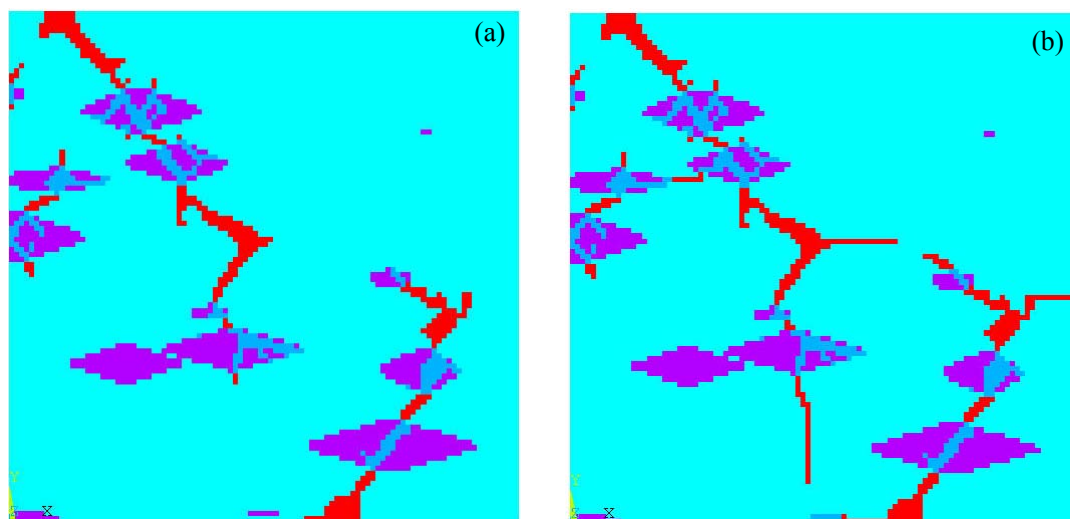


Figure 39: Crack propagation in cyclic loading after (a) 1000 simulated cycles and (b) 6000 simulated cycles.

around the failed CG elements then undergo higher levels of plastic deformation themselves and begin to accumulate damage at higher rates. Coarse grains in close proximity to each other fail first, followed by the UFG matrix between them. Once the initial cracks have coalesced, they work their way through the remaining UFG matrix until fatigue failure occurs.

Three-Dimensional Model

Similarly to the 2D model, cracks nucleate on the edges of CGs, as shown in Figure 40(a). Damage is prevalent throughout every CG element as seen in Figure 40(b). The more damaged UFG elements are not shown in this figure, but their damages are low in comparison to the failing CGs. Again it is the higher levels of plastic strain suffered by the CGs that initiate the crack..

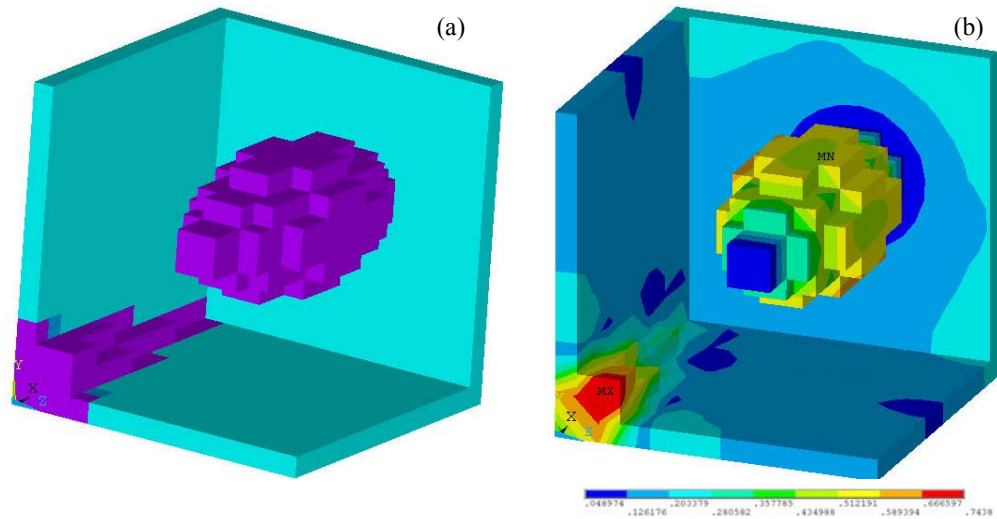


Figure 40: Crack initiation in cyclic loading after 400 simulated cycles; (a) eliminated elements in the microstructure, (b) sites of maximum accumulated damage.

Five FE iteration (1000 simulated cycles) later, a crack has developed through the CG, as shown in Figure 41(a). As expected, the damage levels have increased almost exclusively in the CGs (Figure 41(b)).

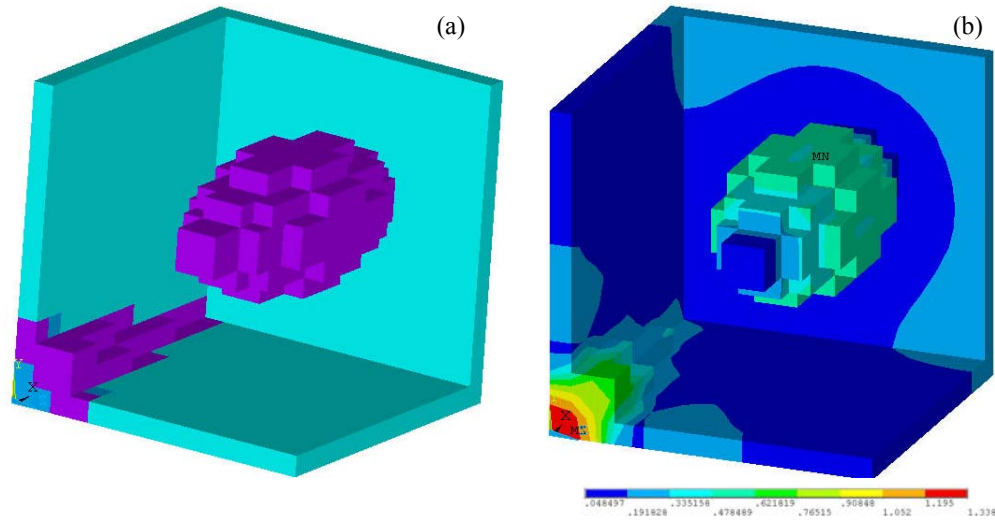


Figure 41: Crack propagation in cyclic loading after 1200 simulated cycles; (a) eliminated elements in the microstructure, (b) sites of maximum accumulated damage.

It takes 400 more simulated cycles before the crack breaches the UFG matrix, as shown in Figure 42(a). Over the next five FE iterations, it grows into the crack seen in Figure 42(b). The three dimensional crack grows outward from the CG in a way impossible to visualize using the 2D model. Unfortunately the level of detail possible in the 3D model limits the number of CGs that the crack can interact with, so essentially only the progression of the crack between two CGs is modeled in three dimensions.

The following FE iteration (2800 simulated cycles) marks the beginnings of failure in the central CG, as shown in Figure 43(a). These failed CG elements coalesce with the growing crack 200 simulated cycles later in Figure 43(b). Enough of the model's elements are eliminated that cascading failure is forthcoming.

And it occurs 400 simulated cycles later, as shown in Figure 44(a). The crack simultaneously moves around through the rest of the central CG before reaching the far corner of the model in Figure 44(b).

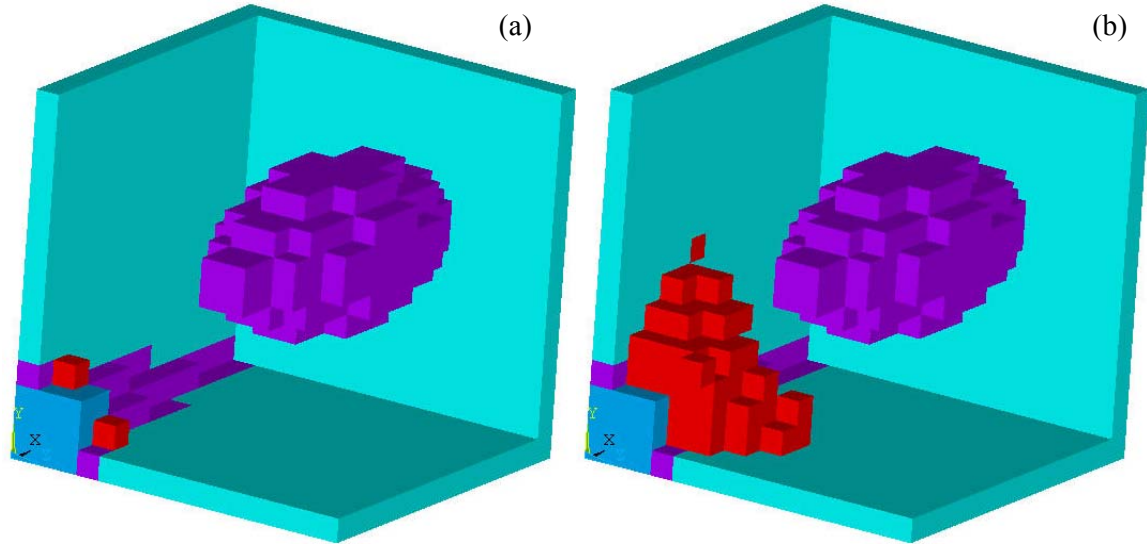


Figure 42: Crack propagation in cyclic loading after (a) 1600 simulated cycles and (b) 2600 simulated cycles.

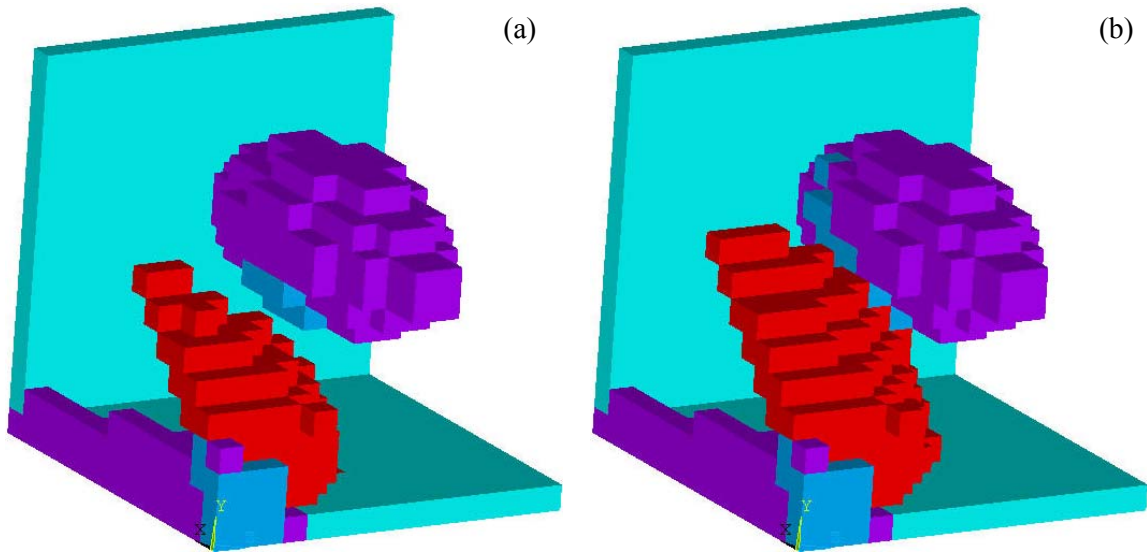


Figure 43: Crack propagation in cyclic loading after (a) 2800 simulated cycles and (b) 3000 simulated cycles.

The 2D and 3D fatigue models agree with and support each other in the same way the 2D and 3D tensile models do. In both, the crack nucleates within a CG at the CG-UFG interface. It progresses through first the CG regions where it nucleated and then through the surround UFG

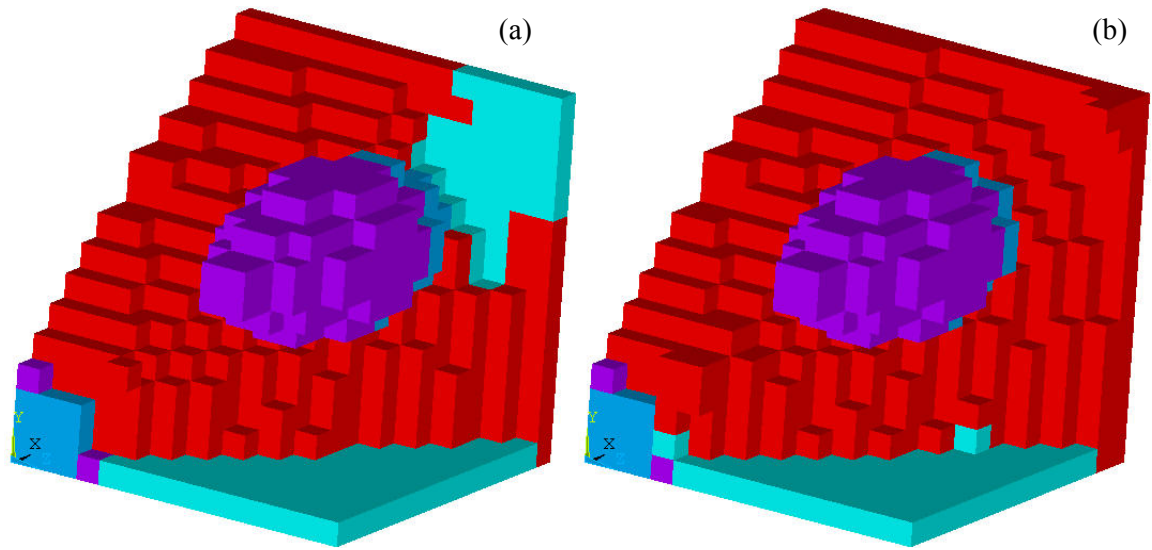


Figure 44: Crack propagation in cyclic loading after (a) 3400 simulated cycles and (b) 3800 simulated cycles.

regions. Ultimately cracks that started in different CGs join together and course through the rest of the material until the model fails.

Appendix B: ANSYS Input Files

Monotonic 2D Input Files

CIPped Global Model

```
!Builds the 2D Tensile global model with CIPped properties
/FILNAME,model,0
/CWD,'C:\ANSYS\2D Tensile CIP Global'

!All dimensions are in millimeters
/PREP7                                !Starts Preprocessor

!Define geometry!
K,1,0.00,0.00,0.00                    !Define keypoint 01 at (0.00,0.00,0.00)
K,2,3.75,0.00,0.00                    !Define keypoint 02 at (3.75,0.00,0.00)
K,3,5.00,1.25,0.00                    !Define keypoint 03 at (5.00,1.25,0.00)
K,4,6.25,0.00,0.00                    !Define keypoint 04 at (6.25,0.00,0.00)
K,5,8.00,0.00,0.00                    !Define keypoint 05 at (8.00,0.00,0.00)
K,6,8.00,3.00,0.00                    !Define keypoint 06 at (8.00,3.00,0.00)
K,7,3.00,3.00,0.00                    !Define keypoint 07 at (3.00,3.00,0.00)
K,8,3.00,0.50,0.00                    !Define keypoint 08 at (3.00,0.50,0.00)
K,9,0.00,0.50,0.00                    !Define keypoint 09 at (0.00,0.50,0.00)

L,1,2                                  !Define line 01 between points 1 & 2
LARC,2,4,3,1.25                       !Define line 02 as arc with rad 1.25 between points 2,3,4
L,4,5                                  !Define line 03 between points 4 & 5
L,5,6                                  !Define line 04 between points 5 & 6
L,6,7                                  !Define line 05 between points 6 & 7
LFILLT,4,5,1.25                       !Define line 06 (fillet) with rad 1.25 between lines 4 & 5
LCOMB,4,6                              !Define line 04 by combining lines 4 & 6 (06 is deleted)
L,7,8                                  !Define line 06 between points 7 & 8
L,8,9                                  !Define line 07 between points 8 & 9
LFILLT,6,7,1.25                       !Define line 08 (fillet) with rad 1.25 between lines 7 & 8
LCOMB,6,8                              !Define line 06 by combining lines 6 and 8 (08 is deleted)
L,9,1                                  !Define line 08 between points 9 and 1
AL,1,2,3,4,5,6,7,8                    !Creates area from lines 1, 2, 3, 4, 5, 6, 7, and 8

!Define global model as 15% cg content from Joshi's model
MPTEMP,,,,,,,,                        !Defines elastic material properties
MPTEMP,1,0
MPDATA,EX,1,,70000                    !Defines Young's Modulus
MPDATA,PRXY,1,,0.33                  !Defines Poisson's Ratio

TB,MISO,1,1,57,0                      !Defines multilinear plastic properties
TBTEMP,0
TBPT,,0.0079,553.0
TBPT,,0.0080,560.0
TBPT,,0.0081,563.4
TBPT,,0.0082,565.1
TBPT,,0.0083,566.8
TBPT,,0.0084,568.4
TBPT,,0.0085,570.1
TBPT,,0.0086,571.7
TBPT,,0.0087,573.3
TBPT,,0.0088,574.8
TBPT,,0.0089,576.4
TBPT,,0.0090,577.9
TBPT,,0.0091,579.4
TBPT,,0.0092,580.9
TBPT,,0.0093,582.3
TBPT,,0.0094,583.8
TBPT,,0.0095,585.2
TBPT,,0.0096,586.6
TBPT,,0.0097,588.0
TBPT,,0.0098,589.4
```

```

TBPT,,0.0099,590.7
TBPT,,0.0100,592.1
TBPT,,0.0102,594.7
TBPT,,0.0104,597.2
TBPT,,0.0106,599.7
TBPT,,0.0108,602.1
TBPT,,0.0110,604.5
TBPT,,0.0112,606.8
TBPT,,0.0114,609.0
TBPT,,0.0116,611.2
TBPT,,0.0118,613.3
TBPT,,0.0120,615.3
TBPT,,0.0125,620.3
TBPT,,0.0130,624.9
TBPT,,0.0135,629.2
TBPT,,0.0140,633.2
TBPT,,0.0145,637.0
TBPT,,0.0150,640.5
TBPT,,0.0155,643.8
TBPT,,0.0160,646.9
TBPT,,0.0165,649.8
TBPT,,0.0170,652.5
TBPT,,0.0175,655.0
TBPT,,0.0180,657.4
TBPT,,0.0190,661.7
TBPT,,0.0200,665.5
TBPT,,0.0210,668.8
TBPT,,0.0220,671.7
TBPT,,0.0230,674.2
TBPT,,0.0240,676.4
TBPT,,0.0250,678.3
TBPT,,0.0300,685.0
TBPT,,0.0350,688.4
TBPT,,0.0400,690.1
TBPT,,0.0450,691.0
TBPT,,0.0500,691.5
TBPT,,0.1000,692.0

!Last stress/strain data point

ET,1,PLANE183
KEYOPT,1,1,0
KEYOPT,1,3,0
KEYOPT,1,6,0
KEYOPT,1,10,0

!Defines element to mesh with
!Indicates a full integration
!Indicates plane strain
!Indicates pure displacement
!Indicates no user defined initial stress

LESIZE,08,,,50
LESIZE,01,0.01
LESIZE,07,0.01
LESIZE,02,,,60
LESIZE,06,,,60
LESIZE,03,,,7
LESIZE,04,,,12
LESIZE,05,,,15

!Mesh the model
MSHKEY,0
CM,_Y,AREA
ASEL,,,1
CM,_Y1,AREA
CHKMSH,'AREA'
CMSEL,S,_Y
AMESH,_Y1
CMDELE,_Y
CMDELE,_Y1
CMDELE,_Y2

ALLSEL,ALL
FINISH
SAVE,,,ALL

!Reselect all nodes
!Exit the preprocessor
!Save the model and the constraints

```


CIPped Global Solution

```

!Start 2D Global Tensile Analysis
!Requires geometry, elements, and material properties be input previously
/SOLU                                !Start Solver
OUTRES,ERASE                          !Erase output files
FINISH                                !Exit Solver

*DIM,Fname,CHAR,30,1,1,,,            !Create a vector name Fname
*CREATE,ansuitmp                      !Create an input macro file
*VREAD,Fname,'Fname','txt',' ',IJK,30,1,1,, !Read from Fname.txt to Fname vector
(A)                                  !Read input in character format
*END                                  !Close the macro file
/INPUT,ansuitmp                      !Switch the input file for following commands
FINISH                                !Finish the file input

!Apply boundary conditions
/PREP7                                !Start the preprocessor
!_____!
!Apply symmetry to x y and z boundaries at 0
NSEL,S,LOC,X,0.0                    !Select nodes at x = 0.00
DSYM,SYMM,X                          !Apply symmetry conditions
NSEL,S,LOC,Y,0.0                    !Select nodes at y = 0.00
DSYM,SYMM,Y                          !Apply symmetry conditions
ALLSEL,ALL                          !Reselect all nodes

!Solve using incremental steps starting at 80% of the yield strength and ending at the
!ultimate tensile strength
yield_str = 560.0                    !Define the yield strength
ulten_str = 690.0                    !Define the UTS
gage_len = 3.00                     !Define the gage length
x_disp_ini = 560.0*gage_len/70000.0 !Define the starting displacement
x_disp_fin = 690.0*5.0/70000.0      !Define the final displacement
num_steps = 30.0                     !Define the number of steps
step_size = (x_disp_fin - x_disp_ini)/num_steps !Define the step size

*DO,q,1,30,1
FINISH                                !Exit the solver
/FILNAME,Fname(q,1,1),1             !Create output filename
/PREP7                                !Start the preprocessor
LSEL,S,LINE,,2                       !Select the pinhole
NSLL,,1                              !Select the nodes on the pinhole
NSEL,R,LOC,X,5.0,6.25               !Select the upper pinhole surface
DDELE,ALL,UX                         !Delete existing displacements
x_disp = x_disp_ini + q*step_size    !Define current load value
D,ALL,UX,x_disp                     !Displace the pinhole nodes
ALLSEL,ALL                          !Reselect everything
FINISH                                !Exit the preprocessor

!Start load steps
/SOLU                                !Start the solver
NSUBST,100                          !Specify 100 substeps
SOLVE                                !Solve the load step
FINISH                                !Exit the solver

ALLSEL,ALL                          !Reselect all nodes
SAVE,,,ALL                          !Save all database information
FINISH                                !Exit save

*ENDDO

```

CIPped Local Model

```

!Tensile local model containing geometry, material, and element information for CIPped
!material - All dimensions are in millimeters
/FILNAME,model,0
/CWD,'C:\ANSYS\2D Tensile CIP Local 200x200'

/PREP7                                !Starts preprocessor
!Define geometry
K,1,0.00,0.00,0.0                    !Define keypoint 1 at (0.00,0.00,0.0) mm
K,2,0.01,0.00,0.0                    !Define keypoint 2 at (0.01,0.00,0.0) mm
K,3,0.01,0.01,0.0                    !Define keypoint 3 at (0.01,0.01,0.0) mm
K,4,0.00,0.01,0.0                    !Define keypoint 4 at (0.00,0.01,0.0) mm
A,1,2,3,4                            !Define area 1 from points 1-4

!Defines elastic material properties for material 1: 0.0% Coarse Grain Al-5083
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,70000                    !Defines Young's Modulus
MPDATA,PRXY,1,,0.33                  !Defines Poisson's Ratio

!Defines elastic material properties for material 2: 100% Coarse Grain Al-5083
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,2,,70000                    !Defines Young's Modulus
MPDATA,PRXY,2,,0.33                  !Defines Poisson's Ratio

!Defines plastic material properties for material 1: 0.0% Coarse Grain Al-5083
TB,MISO,1,1,78,0                      !Defines multilinear plastic properties
TBTEMP,0                             !for 0% Coarse Grain Al-5083 using
TBPT,,0.0080,560.0                    !Joshi's Voce Plasticity Model
TBPT,,0.0081,567.0
TBPT,,0.0082,574.0
TBPT,,0.0083,579.0
TBPT,,0.0084,581.5
TBPT,,0.0085,583.1
TBPT,,0.0086,584.6
TBPT,,0.0087,586.0
TBPT,,0.0088,587.5
TBPT,,0.0089,589.0
TBPT,,0.0090,590.4
TBPT,,0.0091,591.8
TBPT,,0.0092,593.2
TBPT,,0.0093,594.6
TBPT,,0.0094,596.0
TBPT,,0.0095,597.3
TBPT,,0.0096,598.6
TBPT,,0.0097,600.0
TBPT,,0.0098,601.3
TBPT,,0.0099,602.5
TBPT,,0.0100,603.8
TBPT,,0.0101,605.1
TBPT,,0.0102,606.3
TBPT,,0.0103,607.5
TBPT,,0.0104,608.8
TBPT,,0.0105,610.0
TBPT,,0.0106,611.1
TBPT,,0.0107,612.3
TBPT,,0.0108,613.5
TBPT,,0.0109,614.6
TBPT,,0.0110,615.7
TBPT,,0.0111,616.9
TBPT,,0.0112,618.0
TBPT,,0.0113,619.0
TBPT,,0.0114,620.1
TBPT,,0.0115,621.2
TBPT,,0.0116,622.2

```

```

TBPT,,0.0117,623.3
TBPT,,0.0118,624.3
TBPT,,0.0119,625.3
TBPT,,0.0120,626.3
TBPT,,0.0122,628.3
TBPT,,0.0124,630.2
TBPT,,0.0126,632.1
TBPT,,0.0128,633.9
TBPT,,0.0130,635.7
TBPT,,0.0132,637.5
TBPT,,0.0134,639.2
TBPT,,0.0136,640.8
TBPT,,0.0138,642.5
TBPT,,0.0140,644.0
TBPT,,0.0145,647.9
TBPT,,0.0150,651.4
TBPT,,0.0155,654.8
TBPT,,0.0160,658.0
TBPT,,0.0165,661.0
TBPT,,0.0170,663.9
TBPT,,0.0175,666.5
TBPT,,0.0180,669.0
TBPT,,0.0185,671.4
TBPT,,0.0190,673.6
TBPT,,0.0195,675.7
TBPT,,0.0200,677.7
TBPT,,0.0210,681.3
TBPT,,0.0220,684.6
TBPT,,0.0230,687.4
TBPT,,0.0240,690.0
TBPT,,0.0250,692.2
TBPT,,0.0260,694.2
TBPT,,0.0270,696.0
TBPT,,0.0280,697.5
TBPT,,0.0290,698.9
TBPT,,0.0300,700.2
TBPT,,0.0350,704.6
TBPT,,0.0400,707.0
TBPT,,0.0450,708.4
TBPT,,0.0500,709.1
TBPT,,0.1000,710.0                                !Last stress/strain data point

!Define plastic material properties for material 2: 100% Coarse Grain Al-5083
TB,MISO,2,1,85,0                                !Defines multilinear plastic properties
TBTEMP,0                                          !for 100% Coarse Grain Al-5083 using
TBPT,,0.0040,280.0                                !Joshi's Voce Plasticity Model
TBPT,,0.0041,287.0
TBPT,,0.0042,293.0
TBPT,,0.0043,295.4
TBPT,,0.0044,295.7
TBPT,,0.0045,296.1
TBPT,,0.0046,296.5
TBPT,,0.0047,296.8
TBPT,,0.0048,297.2
TBPT,,0.0049,297.6
TBPT,,0.0050,297.9
TBPT,,0.0051,298.3
TBPT,,0.0052,298.7
TBPT,,0.0053,299.0
TBPT,,0.0054,299.4
TBPT,,0.0055,299.7
TBPT,,0.0056,300.1
TBPT,,0.0057,300.5
TBPT,,0.0058,300.8
TBPT,,0.0059,301.2
TBPT,,0.0060,301.5
TBPT,,0.0062,302.2
TBPT,,0.0064,302.9

```

```

TBPT,,0.0066,303.6
TBPT,,0.0068,304.3
TBPT,,0.0070,305.0
TBPT,,0.0072,305.7
TBPT,,0.0074,306.4
TBPT,,0.0076,307.0
TBPT,,0.0078,307.7
TBPT,,0.0080,308.4
TBPT,,0.0082,309.0
TBPT,,0.0084,309.7
TBPT,,0.0086,310.4
TBPT,,0.0088,311.0
TBPT,,0.0090,311.7
TBPT,,0.0092,312.3
TBPT,,0.0094,312.9
TBPT,,0.0096,313.6
TBPT,,0.0098,314.2
TBPT,,0.0100,314.6
TBPT,,0.0102,315.5
TBPT,,0.0104,316.1
TBPT,,0.0106,316.7
TBPT,,0.0108,317.3
TBPT,,0.0110,318.0
TBPT,,0.0112,318.6
TBPT,,0.0114,319.2
TBPT,,0.0116,319.8
TBPT,,0.0118,320.4
TBPT,,0.0120,321.0
TBPT,,0.0125,322.4
TBPT,,0.0130,323.9
TBPT,,0.0135,325.3
TBPT,,0.0140,326.7
TBPT,,0.0145,328.1
TBPT,,0.0150,329.5
TBPT,,0.0155,330.9
TBPT,,0.0160,332.2
TBPT,,0.0165,333.5
TBPT,,0.0170,334.8
TBPT,,0.0175,336.1
TBPT,,0.0180,337.4
TBPT,,0.0190,339.8
TBPT,,0.0200,342.2
TBPT,,0.0210,344.6
TBPT,,0.0220,346.8
TBPT,,0.0230,349.0
TBPT,,0.0240,351.2
TBPT,,0.0250,353.2
TBPT,,0.0275,358.2
TBPT,,0.0300,362.8
TBPT,,0.0325,367.1
TBPT,,0.0350,371.1
TBPT,,0.0375,374.8
TBPT,,0.0400,378.3
TBPT,,0.0425,381.5
TBPT,,0.0450,384.5
TBPT,,0.0475,387.3
TBPT,,0.0500,389.9
TBPT,,0.0550,394.5
TBPT,,0.0600,398.6
TBPT,,0.0700,405.2
TBPT,,0.0800,410.1
TBPT,,0.1000,416.6
!Last stress/strain data point

!Create dummy material 3 to change killed ufg elements to (material 1)
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,3,,70000
MPDATA,PRXY,3,,0.33
!Defines Young's Modulus
!Defines Poisson's Ratio

```

```

!Create dummy material 4 to change killed cg elements to (material 2)
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,4,,70000                                !Defines Young's Modulus
MPDATA,PRXY,4,,0.33                                !Defines Poisson's Ratio

!Define element to mesh with
ET,1,PLANE183                                        !Defines SOLID186 element
KEYOPT,1,1,0                                        !Indicates a full integration
KEYOPT,1,3,0                                        !Indicates plane strain
KEYOPT,1,6,0                                        !Indicates pure displacement
KEYOPT,1,10,0                                       !Indicates no user defined initial stress
LSEL,ALL                                             !Selects all boundary lines
numb = 200                                           !Specifies 'numb' elements divisions per line
LESIZE,ALL,,numb                                    !Create 100 element division per line
ee=0.01/numb                                         !Computes the element size

!Mesh the model
MSHAPE,0,2D
MSHKEY,1
CM,_Y,AREA
ASEL,,,,,1
CM,_Y1,AREA
CHKMSH,'AREA'
CMSEL,S,_Y
AMESH,_Y1
CMDELE,_Y
CMDELE,_Y1
CMDELE,_Y2

/PREP7                                              !Start the preprocessor

nn=30                                              !Specifies how many coarse grains to create
!Define random locations of the centers of the coarse grains with the UFG domain
*DIM,XYZ,ARRAY,nn,3                                !Defines XYZ array with 3 elements (x,y,z)
*DIM,RL1,ARRAY,nn,3                                !Defines RL1 array with 3 elements (R,+L,-L)
*DO,ii,1,nn,1                                       !Loops through and creates the required
number of volumes
    !Define the spatial locations of the 'center' of each coarse grain
    *VFILL,XYZ(ii,1),RAND,-0.0010,0.0090           !Puts a random number in 1st element of XYZ
    *VFILL,XYZ(ii,2),RAND,-0.0010,0.0090           !Puts a random number in 2nd element of XYZ
    *VFILL,XYZ(ii,3),RAND,-0.0005,0.0005           !Puts a random number in 3rd element of XYZ
    *VFILL,RL1(ii,1),RAND, 0.0002,0.0006           !Puts a random number in 1st element of RL1
    *VFILL,RL1(ii,2),RAND, 0.0010,0.0020           !Puts a random number in 2nd element of RL1
    *VFILL,RL1(ii,3),RAND, 0.0010,0.0020           !Puts a random number in 3rd element of RL1
*ENDDO

!Define random sized coarse grains at the random locations
*DO,ii,1,nn,1
    /VIEW,1,1,,                                     !Changes the view to the global yz-plane
    /ANG,1
    /REP,FAST

    !Moves the origin of the working plane to XY(1),XY(2),0 and aligns the new
    !z-direction with the global x-direction
    WPLANE,1,XYZ(ii,1),XYZ(ii,2),XYZ(ii,3),XYZ(ii,1),XYZ(ii,2),XYZ(ii,3)-0.1
    !Changes the coordinate system to cylindrical coordinates
    CSWPLA,11,1

    !Pick nodes inside volume defined by XYZ and RL1
    ESEL,NONE
    NSEL,NONE
    *DO,zz,(ee/2),RL1(ii,2),ee
        rad = RL1(ii,1) - ((RL1(ii,1) - ee)/RL1(ii,2))*zz
        NSEL,A,LOC,Z,(zz - ee),(zz + ee)
        NSEL,R,LOC,X,0.0,rad
        ESLN,A,1
    *DOEND

```

```

        EMODIF,ALL,MAT,2
*ENDDO

ESEL,NONE
NSEL,NONE
*DO,zz,-(ee/2),-RL1(ii,3),-ee
    rad = RL1(ii,1) + ((RL1(ii,1) - ee)/RL1(ii,3))*zz
    NSEL,A,LOC,Z,(zz - ee),(zz + ee)
    NSEL,R,LOC,X,0.0,rad
    ESLN,A,1
    EMODIF,ALL,MAT,2
*ENDDO
*ENDDO

ESEL,S,MAT,,2
*GET,CG,ELEM,,COUNT
ESEL,ALL
*GET,TOT,ELEM,,COUNT
cgfrac = CG/TOT

/PNUM,KP,0
/PNUM,LINE,0
/PNUM,AREA,0
/PNUM,VOLU,0
/PNUM,NODE,0
/PNUM,TABN,0
/PNUM,SVAL,0
/NUMBER,1
/PNUM,MAT,1
EPLLOT
/VIEW,1,,,1                !Changes the view to an isometric view
/ANG,1
/REP,FAST
/AUTO,1
/REP,FAST

ALLSEL,ALL
CSYS,0
FINISH
SAVE,,,ALL
FINISH

```

CIPped Local Solution

```

!Start 2D Local Tensile Analysis
!Requires geometry, elements, and material properties be input previously
/SOLU                        !Start solver
OUTRES,ERASE                !Erase output files
FINISH                      !Exit solver
*DIM,Fname,CHAR,30,1,1,,,   !Create a vector named Fname
*CREATE,ansuitmp            !Create an input macro file
*VREAD,Fname,'Fname','txt',' ',IJK,30,1,1,, !Read from Fname.txt to Fname vector
(A)                         !Read input in character format
*END                        !Close the macro file
/INPUT,ansuitmp             !Switch the input file for following commands
FINISH                      !Finish the file input

!Apply boundary conditions and input load file
/PREP7                      !Start the preprocessor
!
!Apply symmetry to x, y, and z boundaries at 0
NSEL,S,LOC,X,0.0            !Select nodes at x = 0.0
DSYM,SYMM,X                 !Apply symmetry condition
NSEL,S,LOC,Y,0.0            !Select nodes at y = 0.0
DSYM,SYMM,Y                 !Apply symmetry condition
ALLSEL,ALL                  !Reselect all nodes

```



```

EMODIF,ALL,MAT,3          !Changes killed UFG elements to material 3
FINISH                    !Exit the preprocessor

SAVE,,,,ALL               !Save the current solution to b,c,d,...,txt
FINISH                    !Exit the save
*ENDDO

```

HIPped Global Model

```

!Builds the 2D global model with HIPped properties
/FILNAME,model,0
/CWD,'C:\ANSYS\2D Tensile HIP Global'

!All dimensions are in millimeters
/PREP7                    !Starts ANSYS

!Define geometry!
K,1,0.00,0.00,0.00        !Define keypoint 01 at (0.00,0.00,0.00)
K,2,3.75,0.00,0.00        !Define keypoint 02 at (3.75,0.00,0.00)
K,3,5.00,1.25,0.00        !Define keypoint 03 at (5.00,1.25,0.00)
K,4,6.25,0.00,0.00        !Define keypoint 04 at (6.25,0.00,0.00)
K,5,8.00,0.00,0.00        !Define keypoint 05 at (8.00,0.00,0.00)
K,6,8.00,3.00,0.00        !Define keypoint 06 at (8.00,3.00,0.00)
K,7,3.00,3.00,0.00        !Define keypoint 07 at (3.00,3.00,0.00)
K,8,3.00,0.50,0.00        !Define keypoint 08 at (3.00,0.50,0.00)
K,9,0.00,0.50,0.00        !Define keypoint 09 at (0.00,0.50,0.00)

L,1,2                     !Define line 01 between points 1 and 2
LARC,2,4,3,1.25           !Define line 02 as arc with rad 1.25 between points 2,3,4
L,4,5                     !Define line 03 between points 4 and 5
L,5,6                     !Define line 04 between points 5 and 6
L,6,7                     !Define line 05 between points 6 and 7
LFILLT,4,5,1.25           !Define line 06 (fillet) with rad 1.25 between lines 4 & 5
LCOMB,4,6                 !Define line 04 by combining lines 4 and 6 (06 is deleted)
L,7,8                     !Define line 06 between points 7 and 8
L,8,9                     !Define line 07 between points 8 and 9
LFILLT,6,7,1.25           !Define line 08 (fillet) with rad 1.25 between lines 7 & 8
LCOMB,6,8                 !Define line 06 by combining lines 6 and 8 (08 is deleted)
L,9,1                     !Define line 08 between points 9 and 1
AL,1,2,3,4,5,6,7,8        !Creates area from lines 1, 2, 3, 4, 5, 6, 7, and 8

!Define global model as 15% cg content from Joshi's model
MPTEMP,,,,,,,,            !Defines elastic material properties
MPTEMP,1,0
MPDATA,EX,1,,72500        !Defines Young's Modulus
MPDATA,PRXY,1,,0.33       !Defines Poisson's Ratio

TB,MISO,1,1,73,0          !Defines multilinear plastic properties
TBTEMP,0
TBPT,,0.0052,377.000
TBPT,,0.0053,380.100
TBPT,,0.0054,381.077
TBPT,,0.0055,382.040
TBPT,,0.0056,382.990
TBPT,,0.0057,383.927
TBPT,,0.0058,384.850
TBPT,,0.0059,385.761
TBPT,,0.0060,386.659
TBPT,,0.0061,387.544      !10 data points
TBPT,,0.0062,388.417
TBPT,,0.0063,389.278
TBPT,,0.0064,390.126
TBPT,,0.0065,390.963
TBPT,,0.0066,391.789
TBPT,,0.0067,392.602
TBPT,,0.0068,393.405

```



```

TBPT,,0.0069,394.196
TBPT,,0.0070,394.976
TBPT,,0.0071,395.745      !20 data points
TBPT,,0.0072,396.503
TBPT,,0.0073,397.251
TBPT,,0.0074,397.989
TBPT,,0.0075,398.716
TBPT,,0.0076,399.433
TBPT,,0.0077,400.140
TBPT,,0.0078,400.837
TBPT,,0.0079,401.524
TBPT,,0.0080,402.202
TBPT,,0.0082,403.529      !30 data points
TBPT,,0.0084,404.219
TBPT,,0.0086,406.074
TBPT,,0.0088,407.294
TBPT,,0.0090,408.480
TBPT,,0.0092,409.633
TBPT,,0.0094,410.754
TBPT,,0.0096,411.844
TBPT,,0.0098,412.904
TBPT,,0.0100,413.934
TBPT,,0.0105,416.386      !40 data points
TBPT,,0.0110,418.672
TBPT,,0.0115,420.803
TBPT,,0.0120,422.789
TBPT,,0.0125,424.641
TBPT,,0.0130,426.366
TBPT,,0.0135,427.974
TBPT,,0.0140,429.473
TBPT,,0.0145,430.871
TBPT,,0.0150,432.173
TBPT,,0.0155,433.387      !50 data points
TBPT,,0.0160,434.519
TBPT,,0.0165,435.573
TBPT,,0.0170,436.556
TBPT,,0.0175,437.473
TBPT,,0.0180,438.327
TBPT,,0.0190,439.865
TBPT,,0.0200,441.201
TBPT,,0.0210,442.362
TBPT,,0.0220,443.371
TBPT,,0.0230,444.247      !60 data points
TBPT,,0.0240,445.009
TBPT,,0.0250,445.670
TBPT,,0.0300,447.882
TBPT,,0.0350,448.977
TBPT,,0.0400,449.519
TBPT,,0.0450,449.787
TBPT,,0.0500,449.918
TBPT,,0.0600,450.018
TBPT,,0.0700,450.042
TBPT,,0.0800,450.048      !70 data points
TBPT,,0.0900,450.050
TBPT,,0.1000,450.050
TBPT,,0.1100,450.050      !Last stress/strain data point

ET,1,PLANE183              !Defines element to mesh with
KEYOPT,1,1,0              !Indicates a full integration
KEYOPT,1,3,0              !Indicates plane strain
KEYOPT,1,6,0              !Indicates pure displacement
KEYOPT,1,10,0             !Indicates no user defined initial stress

LESIZE,08,,50
LESIZE,01,0.01
LESIZE,07,0.01
LESIZE,02,,60
LESIZE,06,,60

```

```

LESIZE,03,,,7
LESIZE,04,,,12
LESIZE,05,,,15

!Mesh the model
MSHKEY,0
CM,_Y,AREA
ASEL,,,,1
CM,_Y1,AREA
CHKMSH,'AREA'
CMSEL,S,_Y
AMESH,_Y1
CMDELE,_Y
CMDELE,_Y1
CMDELE,_Y2

ALLSEL,ALL                !Reselect all nodes
FINISH                    !Exit the preprocessor
SAVE,,,,ALL               !Save the model and the constraints

```

HIPped Global Solution

```

!Start 2D HIPped Tensile Analysis
!Requires geometry, elements, and material properties be input previously
/SOLU                      !Start Solver
OUTRES,ERASE              !Erase output files
FINISH                    !Exit Solver

*DIM,Fname,CHAR,30,1,1,,, !Create a vector name Fname
*CREATE,ansuitmp          !Create an input macro file
*VREAD,Fname,'Fname','txt',' ',IJK,30,1,1,, !Read from Fname.txt to Fname vector
(A)                      !Read input in character format
*END                      !Close the macro file
/INPUT,ansuitmp           !Switch the input file for following commands
FINISH                   !Finish the file input

!Apply boundary conditions
/PREP7                    !Start the preprocessor
!_____!
!Apply symmetry to x y and z boundaries at 0
NSSEL,S,LOC,X,0.0        !Select nodes at x = 0.00
DSYM,SYMM,X              !Apply symmetry conditions
NSSEL,S,LOC,Y,0.0        !Select nodes at y = 0.00
DSYM,SYMM,Y              !Apply symmetry conditions
ALLSEL,ALL               !Reselect all nodes

!Solve using incremental steps starting at 80% of the yield strength and ending at the
!ultimate tensile strength
yield_str = 380.0        !Define the yield strength
ulten_str = 450.0        !Define the UTS
gage_len = 3.00         !Define the gage length
x_disp_ini = 0.8*380.0*gage_len/72500.0 !Define the starting displacement
x_disp_fin = 450.0*gage_len/72500.0     !Define the final displacement
num_steps = 30.0         !Define the number of steps
step_size = (x_disp_fin - x_disp_ini)/num_steps !Define the step size

*DO,q,1,30,1
FINISH                    !Exit the solver
/FILNAME,Fname(q,1,1),1 !Create output filename
/PREP7                   !Start the preprocessor
LSEL,S,LINE,,2           !Select the pinhole
NSLL,,1                  !Select the nodes on the pinhole
NSSEL,R,LOC,X,5.0,6.25   !Select the upper pinhole surface
DDELE,ALL,UX             !Delete existing displacements

```

```

x_disp = x_disp_ini + q*step_size      !Define current load value
D,ALL,UX,x_disp                        !Displace the pinhole nodes
ALLSEL,ALL                             !Reselect everything
FINISH                                 !Exit the preprocessor

!Start load steps
/SOLU                                  !Start the solver
NSUBST,100                             !Specify 100 substeps
SOLVE                                  !Solve the load step
FINISH                                 !Exit the solver

ALLSEL,ALL                             !Reselect all nodes
SAVE,,,ALL                             !Save all database information
FINISH                                 !Exit save

*ENDDO

```

HIPped Local Model

```

!Fatigue model containing geometry, material, and element information
!All dimensions are in millimeters
/FILNAME,model,0
/CWD,'C:\ANSYS\2D Tensile HIP Local 200x200'

/PREP7                                !Starts preprocessor
!Define geometry
K,1,0.00,0.00,0.0                     !Define keypoint 1 at (0.00,0.00,0.0) mm
K,2,0.01,0.00,0.0                     !Define keypoint 2 at (0.01,0.00,0.0) mm
K,3,0.01,0.01,0.0                     !Define keypoint 3 at (0.01,0.01,0.0) mm
K,4,0.00,0.01,0.0                     !Define keypoint 4 at (0.00,0.01,0.0) mm
A,1,2,3,4                             !Define area 1 from points 1-4

!Defines elastic material properties for material 1: 0.0% Coarse Grain Al-5083
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,72500                    !Defines Young's Modulus
MPDATA,PRXY,1,,0.33                  !Defines Poisson's Ratio

!Defines elastic material properties for material 2: 100% Coarse Grain Al-5083
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,2,,72500                    !Defines Young's Modulus
MPDATA,PRXY,2,,0.33                  !Defines Poisson's Ratio

!Defines plastic material properties for material 1: 0.0% Coarse Grain Al-5083
TB,MISO,1,1,93,0                      !Defines multilinear plastic properties
TBTEMP,0                              !for 0% Coarse Grain Al-5083 using
TBPT,,0.0060,435.000                  !Joshi's Voce Plasticity Model
TBPT,,0.0061,441.000
TBPT,,0.0062,441.485
TBPT,,0.0063,441.965
TBPT,,0.0064,442.438
TBPT,,0.0065,442.907
TBPT,,0.0066,443.369
TBPT,,0.0067,443.826
TBPT,,0.0068,444.278
TBPT,,0.0069,444.725
TBPT,,0.0070,445.166                  !10 data points
TBPT,,0.0071,445.602
TBPT,,0.0072,446.032
TBPT,,0.0073,446.458
TBPT,,0.0074,446.879
TBPT,,0.0075,447.294
TBPT,,0.0076,447.705
TBPT,,0.0077,448.111
TBPT,,0.0078,448.512

```

```

TBPT,,0.0079,448.908
TBPT,,0.0080,449.230
TBPT,,0.0081,449.687
TBPT,,0.0082,450.069
TBPT,,0.0083,450.447
TBPT,,0.0084,450.820
TBPT,,0.0085,451.189
TBPT,,0.0086,451.554
TBPT,,0.0087,451.914
TBPT,,0.0088,452.270
TBPT,,0.0089,452.622
TBPT,,0.0090,452.970
TBPT,,0.0091,453.313
TBPT,,0.0092,453.653
TBPT,,0.0093,453.988
TBPT,,0.0094,454.320
TBPT,,0.0095,454.647
TBPT,,0.0096,454.971
TBPT,,0.0097,455.291
TBPT,,0.0098,455.607
TBPT,,0.0099,455.919
TBPT,,0.0100,456.228
TBPT,,0.0102,456.834
TBPT,,0.0104,457.427
TBPT,,0.0106,458.005
TBPT,,0.0108,458.569
TBPT,,0.0110,459.121
TBPT,,0.0112,459.659
TBPT,,0.0114,460.185
TBPT,,0.0116,460.698
TBPT,,0.0118,461.199
TBPT,,0.0120,461.688
TBPT,,0.0122,462.166
TBPT,,0.0124,462.633
TBPT,,0.0126,463.089
TBPT,,0.0128,463.534
TBPT,,0.0130,463.968
TBPT,,0.0132,464.392
TBPT,,0.0134,464.807
TBPT,,0.0136,465.211
TBPT,,0.0138,465.606
TBPT,,0.0140,465.992
TBPT,,0.0145,466.917
TBPT,,0.0150,467.789
TBPT,,0.0155,468.610
TBPT,,0.0160,469.384
TBPT,,0.0165,470.113
TBPT,,0.0170,470.800
TBPT,,0.0175,471.447
TBPT,,0.0180,472.057
TBPT,,0.0185,472.631
TBPT,,0.0190,473.173
TBPT,,0.0195,473.683
TBPT,,0.0200,474.163
TBPT,,0.0210,475.043
TBPT,,0.0220,475.824
TBPT,,0.0230,476.517
TBPT,,0.0240,477.132
TBPT,,0.0250,477.679
TBPT,,0.0260,478.164
TBPT,,0.0270,478.594
TBPT,,0.0280,479.976
TBPT,,0.0290,479.316
TBPT,,0.0300,479.617
TBPT,,0.0350,480.686
TBPT,,0.0400,481.275
TBPT,,0.0450,481.600
TBPT,,0.0500,481.780

```

!20 data points

!30 data points

!40 data points

!50 data points

!60 data points

!70 data points

!80 data points

```

TBPT,,0.0600,481.933
TBPT,,0.0700,481.980
TBPT,,0.0800,481.994      !90 data points
TBPT,,0.0900,481.998
TBPT,,0.1000,481.999
TBPT,,0.1070,482.000      !Last stress/strain data point

!Define plastic material properties for material 2: 100% Coarse Grain Al-5083
TB,MISO,2,1,33,0          !Defines multilinear plastic properties
TBTEMP,0                  !for 100% Coarse Grain Al-5083 using
TBPT,,0.0039,282.750      !Joshi's Voce Plasticity Model
TBPT,,0.0040,286.000
TBPT,,0.0045,286.683
TBPT,,0.0050,287.332
TBPT,,0.0055,287.950
TBPT,,0.0060,288.538
TBPT,,0.0065,289.067
TBPT,,0.0070,289.628
TBPT,,0.0075,290.134
TBPT,,0.0080,290.616      !10 data points
TBPT,,0.0085,291.073
TBPT,,0.0090,291.509
TBPT,,0.0095,291.923
TBPT,,0.0100,292.316
TBPT,,0.0110,293.048
TBPT,,0.0120,293.709
TBPT,,0.0130,294.308
TBPT,,0.0140,294.850
TBPT,,0.0150,295.340
TBPT,,0.0160,295.783      !20 data points
TBPT,,0.0170,296.185
TBPT,,0.0180,296.548
TBPT,,0.0190,296.876
TBPT,,0.0200,297.173
TBPT,,0.0300,298.960
TBPT,,0.0400,299.617
TBPT,,0.0500,299.859
TBPT,,0.0600,299.948
TBPT,,0.0700,299.981
TBPT,,0.0800,299.993      !30 data points
TBPT,,0.0900,299.997
TBPT,,0.1000,299.999
TBPT,,0.1080,300.000      !Last stress/strain data point

!Create dummy material 3 to change killed ufg elements to (material 1)
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,3,,72500        !Defines Young's Modulus
MPDATA,PRXY,3,,0.33       !Defines Poisson's Ratio

!Create dummy material 4 to change killed cg elements to (material 2)
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,4,,72500        !Defines Young's Modulus
MPDATA,PRXY,4,,0.33       !Defines Poisson's Ratio

!Define element to mesh with
ET,1,PLANE183             !Defines SOLID186 element
KEYOPT,1,1,0              !Indicates a full integration
KEYOPT,1,3,0              !Indicates plane strain
KEYOPT,1,6,0              !Indicates pure displacement
KEYOPT,1,10,0             !Indicates no user defined initial stress
LSEL,ALL                  !Selects all boundary lines
numb = 200                !Specifies 'numb' elements divisions per line
LESIZE,ALL,,numb          !Create 100 element division per line
ee=0.01/numb              !Computes the element size

!Mesh the model

```

```

MSHAPE,0,2D
MSHKEY,1
CM,_Y,AREA
ASEL,,,1
CM,_Y1,AREA
CHKMSH,'AREA'
CMSEL,S,_Y
AMESH,_Y1
CMDELE,_Y
CMDELE,_Y1
CMDELE,_Y2

/PREP7                                !Start the preprocessor
nn=30                                !Specifies how many coarse grains to create
!Define random locations of the centers of the coarse grains with the UFG domain
*DIM,XYZ,ARRAY,nn,3                  !Defines XYZ array with 3 elements (x,y,z)
*DIM,RL1,ARRAY,nn,3                  !Defines RL1 array with 3 elements (R,+L,-L)
*DO,ii,1,nn,1                        !Loops through and creates the required
number of volumes
    !Define the spatial locations of the 'center' of each coarse grain
    *VFILL,XYZ(ii,1),RAND,-0.0010,0.0090    !Puts a random number in 1st element of XYZ
    *VFILL,XYZ(ii,2),RAND,-0.0010,0.0090    !Puts a random number in 2nd element of XYZ
    *VFILL,XYZ(ii,3),RAND,-0.0005,0.0005    !Puts a random number in 3rd element of XYZ
    *VFILL,RL1(ii,1),RAND, 0.0002,0.0006    !Puts a random number in 1st element of RL1
    *VFILL,RL1(ii,2),RAND, 0.0010,0.0020    !Puts a random number in 2nd element of RL1
    *VFILL,RL1(ii,3),RAND, 0.0010,0.0020    !Puts a random number in 3rd element of RL1
*ENDDO

!Define random sized coarse grains at the random locations
*DO,ii,1,nn,1
    /VIEW,1,1,,                      !Changes the view to the global yz-plane
    /ANG,1
    /REP,FAST

    !Moves the origin of the working plane to XY(1),XY(2),0 and aligns the new
    !z-direction with the global x-direction
    WPLANE,1,XYZ(ii,1),XYZ(ii,2),XYZ(ii,3),XYZ(ii,1),XYZ(ii,2),XYZ(ii,3)-0.1
    !Changes the coordinate system to cylindrical coordinates
    CSWPLA,11,1

    !Pick nodes inside volume defined by XYZ and RL1
    ESEL,NONE
    NSEL,NONE
    *DO,zz,(ee/2),RL1(ii,2),ee
        rad = RL1(ii,1) - ((RL1(ii,1) - ee)/RL1(ii,2))*zz
        NSEL,A,LOC,Z,(zz - ee),(zz + ee)
        NSEL,R,LOC,X,0.0,rad
        ESLN,A,1
        EMODIF,ALL,MAT,2
    *ENDDO

    ESEL,NONE
    NSEL,NONE
    *DO,zz,-(ee/2),-RL1(ii,3),-ee
        rad = RL1(ii,1) + ((RL1(ii,1) - ee)/RL1(ii,3))*zz
        NSEL,A,LOC,Z,(zz - ee),(zz + ee)
        NSEL,R,LOC,X,0.0,rad
        ESLN,A,1
        EMODIF,ALL,MAT,2
    *ENDDO
*ENDDO

ESEL,S,MAT,,2
*GET,CG,ELEM,,COUNT
ESEL,ALL
*GET,TOT,ELEM,,COUNT
cgfrac = CG/TOT

```

```

/PNUM,KP,0
/PNUM,LINE,0
/PNUM,AREA,0
/PNUM,VOLU,0
/PNUM,NODE,0
/PNUM,TABN,0
/PNUM,SVAL,0
/NUMBER,1
/PNUM,MAT,1
EPLT
/VIEW,1,,,1          !Changes the view to an isometric view
/ANG,1
/REP,FAST
/AUTO,1
/REP,FAST

ALLSEL,ALL
CSYS,0
FINISH
SAVE,,,ALL
FINISH

```

HIPped Local Solution

```

!Start 2D Tensile HIPped Analysis
!Requires geometry, elements, and material properties be input previously
/SOLU                  !Start solver
OUTRES,ERASE           !Erase output files
FINISH                 !Exit solver
*DIM,Fname,CHAR,30,1,1,, !Create a vector named Fname
*CREATE,ansuitmp        !Create an input macro file
*VREAD,Fname,'Fname','txt',' ',,IJK,30,1,1,, !Read from Fname.txt to Fname vector
(A)                    !Read input in character format
*END                   !Close the macro file
/INPUT,ansuitmp         !Switch the input file for following commands
FINISH                 !Finish the file input

!Apply boundary conditions and input load file
/PREP7                 !Start the preprocessor
!_____!
!Apply symmetry to x, y, and z boundaries at 0
NSEL,S,LOC,X,0.0       !Select nodes at x = 0.0
DSYM,SYMM,X            !Apply symmetry condition
NSEL,S,LOC,Y,0.0       !Select nodes at y = 0.0
DSYM,SYMM,Y            !Apply symmetry condition
ALLSEL,ALL             !Reselect all nodes
FINISH                 !Exit preprocessor

!Read input file to 2D array
*DIM,loads,ARRAY,30,2,1
*CREATE,ansuitmp
*VREAD,loads,'loads','txt',,JIK,2,30,1
(F9.7,2X,F11.8)
*END
/INPUT,ansuitmp
FINISH

*DO,ii,1,2,1
*DO,jj,1,30,1
loads(jj,ii,1)=loads(jj,ii,1)*(6.25/3.00)
*ENDDO
*ENDDO
!_____!

!Iterate through the loads from the global model
!_____!

```

```

*DO,q,1,10,1
FINISH                                !Exit solver
/FILNAME,Fname(q,1,1),1              !Create output filename

/PREP7                                !Start the preprocessor
ALLSEL,ALL                            !Select all nodes and elements

!Apply displacements from local model
NSEL,S,LOC,X,0.01                     !Select nodes at x = 0.01
DDELE,ALL,UX                          !Delete existing displacements
D,ALL,UX,loads(q,1,1)                 !Displace selected nodes
NSEL,S,LOC,Y,0.01                     !Select nodes at y = 0.01
DDELE,ALL,UY                          !Delete existing displacements
D,ALL,UY,loads(q,2,1)                 !Displace selected nodes
ALLSEL,ALL                            !Reselect all nodes
FINISH                                !Exit the preprocessor

!Solve the load step
/SOLU                                  !Start the solver
NSUBST,100                            !Specify 100 substeps
SOLVE                                 !Solve the load step
FINISH                                !Exit the solver

/POST1                                !Start the Postprocessor
ALLSEL,ALL                            !Selects all elements and nodes
ESEL,S,MAT,,2                         !Picks all CG elements
ETABLE,cgs,S,EQV                      !Create element table with equivalent stresses
ESEL,R,ETAB,cgs,290.0                 !Pick elements that have exceeded 390.0 stress

/SOLU                                  !Start the solver
EKILL,ALL                             !Kills selected elements
FINISH                                !Exit the solver

/PREP7                                !Start the preprocessor
EMODIF,ALL,MAT,4                      !Change killed CG elements to material 4
FINISH                                !Exit the preprocessor

/POST1                                !Start the postprocessor
ALLSEL,ALL                            !Selects all elements and nodes
ESEL,S,MAT,,1                         !Picks all UFG elements
ETABLE,ufgs,S,EQV                    !Create element table with the equivalent stresses
ESEL,R,ETAB,ufgs,440.0               !Pick elements that have exceeded 690.0 stress

/SOLU                                  !Start the solver
EKILL,ALL                             !Kills selected elements
FINISH                                !Exit the solver

/PREP7                                !Start the preprocessor
EMODIF,ALL,MAT,3                      !Changes killed UFG elements to material 3
FINISH                                !Exit the preprocessor

SAVE,,,ALL                            !Save the current solution to b,c,d,...,txt
FINISH                                !Exit the save
*ENDDO

```

Monotonic 3D Input Files

CIPped Global Model

```

!Tensile model containing geometry, material, and element information for
!CIPped material - All dimensions are in millimeters
/FILNAME,model,0
/CWD,'C:\ANSYS\3D Tensile CIP Global'
/PREP7                                !Starts ANSYS

```



```

!Define geometry!
K,1,0.00,0.00,0.00      !Define keypoint 01 at (0.00,0.00,0.00)
K,2,3.75,0.00,0.00      !Define keypoint 02 at (3.75,0.00,0.00)
K,3,5.00,1.25,0.00      !Define keypoint 03 at (5.00,1.25,0.00)
K,4,6.25,0.00,0.00      !Define keypoint 04 at (6.25,0.00,0.00)
K,5,8.00,0.00,0.00      !Define keypoint 05 at (8.00,0.00,0.00)
K,6,8.00,3.00,0.00      !Define keypoint 06 at (8.00,3.00,0.00)
K,7,3.00,3.00,0.00      !Define keypoint 07 at (3.00,3.00,0.00)
K,8,3.00,0.50,0.00      !Define keypoint 08 at (3.00,0.50,0.00)
K,9,0.00,0.50,0.00      !Define keypoint 09 at (0.00,0.50,0.00)

L,1,2                    !Define line 01 between points 1 and 2
LARC,2,4,3,1.25          !Define line 02 as arc with rad 1.25 between points 2,3,4
L,4,5                    !Define line 03 between points 4 and 5
L,5,6                    !Define line 04 between points 5 and 6
L,6,7                    !Define line 05 between points 6 and 7
LFILLT,4,5,1.25          !Define line 06 (fillet) with rad 1.25 between lines 4 & 5
LCOMB,4,6                !Define line 04 by combining lines 4 and 6 (06 is deleted)
L,7,8                    !Define line 06 between points 7 and 8
L,8,9                    !Define line 07 between points 8 and 9
LFILLT,6,7,1.25          !Define line 08 (fillet) with rad 1.25 between lines 7 & 8
LCOMB,6,8                !Define line 06 by combining lines 6 and 8 (08 is deleted)
L,9,1                    !Define line 08 between points 9 and 1
AL,1,2,3,4,5,6,7,8      !Creates area from lines 1, 2, 3, 4, 5, 6, 7, and 8
VOFFST,1,0.5            !Creates volume from area 1 by offsetting 0.5 mm in +z-dir

!Define global model as 10% cg content from Joshi's model
MPTEMP,,,,,,,,          !Defines elastic material properties
MPTEMP,1,0
MPDATA,EX,1,,70000      !Defines Young's Modulus
MPDATA,PRXY,1,,0.33     !Defines Poisson's Ratio

TB,MISO,1,1,57,0        !Defines multilinear plastic properties
TBTEMP,0
TBPT,,0.0079,553.0
TBPT,,0.0080,560.0
TBPT,,0.0081,563.4
TBPT,,0.0082,565.1
TBPT,,0.0083,566.8
TBPT,,0.0084,568.4
TBPT,,0.0085,570.1
TBPT,,0.0086,571.7
TBPT,,0.0087,573.3
TBPT,,0.0088,574.8
TBPT,,0.0089,576.4
TBPT,,0.0090,577.9
TBPT,,0.0091,579.4
TBPT,,0.0092,580.9
TBPT,,0.0093,582.3
TBPT,,0.0094,583.8
TBPT,,0.0095,585.2
TBPT,,0.0096,586.6
TBPT,,0.0097,588.0
TBPT,,0.0098,589.4
TBPT,,0.0099,590.7
TBPT,,0.0100,592.1
TBPT,,0.0102,594.7
TBPT,,0.0104,597.2
TBPT,,0.0106,599.7
TBPT,,0.0108,602.1
TBPT,,0.0110,604.5
TBPT,,0.0112,606.8
TBPT,,0.0114,609.0
TBPT,,0.0116,611.2
TBPT,,0.0118,613.3
TBPT,,0.0120,615.3
TBPT,,0.0125,620.3
TBPT,,0.0130,624.9

```

```

TBPT,,0.0135,629.2
TBPT,,0.0140,633.2
TBPT,,0.0145,637.0
TBPT,,0.0150,640.5
TBPT,,0.0155,643.8
TBPT,,0.0160,646.9
TBPT,,0.0165,649.8
TBPT,,0.0170,652.5
TBPT,,0.0175,655.0
TBPT,,0.0180,657.4
TBPT,,0.0190,661.7
TBPT,,0.0200,665.5
TBPT,,0.0210,668.8
TBPT,,0.0220,671.7
TBPT,,0.0230,674.2
TBPT,,0.0240,676.4
TBPT,,0.0250,678.3
TBPT,,0.0300,685.0
TBPT,,0.0350,688.4
TBPT,,0.0400,690.1
TBPT,,0.0450,691.0
TBPT,,0.0500,691.5
TBPT,,0.1000,692.0                                !Last stress/strain data point

ET,2,SOLID186                                !Defines element to mesh with

LSEL,S,LOC,X,0.0
LSEL,A,LINE,,18,23
LESIZE,ALL,,5
LSEL,ALL
LESIZE,01,0.1
LESIZE,09,0.1
LESIZE,07,0.1
LESIZE,15,0.1
LESIZE,02,,18
LESIZE,10,,18
LESIZE,06,,18
LESIZE,14,,18
LESIZE,03,,7
LESIZE,11,,7
LESIZE,04,,12
LESIZE,12,,12
LESIZE,05,,15
LESIZE,13,,15

CM,_Y,VOLU                                !Makes a volume group named _Y
VSEL,,,1                                !Selects volume 1
CM,_Y1,VOLU                                !Makes a volume group named _Y1
CHKMSH,'VOLU'                                !Checks the volume for previous meshes
CMSEL,S,_Y                                !Selects volume 1 to be meshed
VSWEEP,_Y1                                !Meshes volume 1 using swept mapping
CMDELE,_Y                                !Deletes volume group _Y
CMDELE,_Y1                                !Deletes volume group _Y1
CMDELE,_Y2                                !Deletes volume group _Y2

ALLSEL,ALL
FINISH
SAVE,,,ALL

```

CIPped Global Solution

```

!Start 3D CIPped Tensile Analysis
!Requires geometry, elements, and material properties be input previously
/SOLU                                !Start Solver
OUTRES,ERASE                                !Erase output files
FINISH                                !Exit Solver

```

```

*DIM,Fname,CHAR,30,1,1,,,
*CREATE,ansuitmp
*VREAD,Fname,'Fname','txt',' ',,IJK,30,1,1,,
(A)
*END
/INPUT,ansuitmp
commands
FINISH

!Create a vector name Fname
!Create an input macro file
!Read from Fname.txt to Fname vector
!Read input in character format
!Close the macro file
!Switch the input file for following
!Finish the file input

!Apply boundary conditions
/PREP7
!_____!
!Apply symmetry to x y and z boundaries at 0
NSEL,S,LOC,X,0.0
DSYM,SYMM,X
NSEL,S,LOC,Y,0.0
DSYM,SYMM,Y
NSEL,S,LOC,Z,0.0
DSYM,SYMM,Z
ALLSEL,ALL

!Select nodes at x = 0.00
!Apply symmetry conditions
!Select nodes at y = 0.00
!Apply symmetry conditions
!Select nodes at z = 0.00
!Apply symmetry conditions
!Reselect all nodes

!Solve using incremental steps starting at 80% of the yield strength and ending at the
!ultimate tensile strength
yield_str = 560.0
ulten_str = 690.0
gage_len = 6.25
x_disp_ini = 560.0*gage_len/70000.0
x_disp_fin = 690.0*gage_len/70000.0
num_steps = 30.0
step_size = (x_disp_fin - x_disp_ini)/num_steps

!Define the yield strength
!Define the UTS
!Define the gage length
!Define the starting displacement
!Define the final displacement
!Define the number of steps
!Define the step size

*DO,q,1,num_steps,1
FINISH
/FILNAME,Fname(q,1,1),1
/PREP7
ASEL,S,AREA,,4
NSLA,,1
pinhole
NSEL,R,LOC,X,5.0,6.25
DDELE,ALL,UX
x_disp = x_disp_ini + q*step_size
D,ALL,UX,x_disp
ALLSEL,ALL
FINISH

!Exit the solver
!Create output filename
!Start the preprocessor
!Select the pinhole surface
!Select the nodes associated with the
!Select the upper pinhole surface
!Delete existing displacements
!Define current load value
!Displace the pinhole nodes
!Reselect everything
!Exit the preprocessor

!Start load steps
/SOLU
NSUBST,100
SOLVE
FINISH

!Start the solver
!Specify 100 substeps
!Solve the load step
!Exit the solver

ALLSEL,ALL
SAVE,,,ALL
FINISH

!Reselect all nodes
!Save all database information
!Exit save

*ENDDO

```

CIPped Local Model

```

!Tensile model containing geometry, material, and element information for
!CIPped material - All dimensions are in millimeters
/FILNAME,model,0
/CWD,'C:\ANSYS\3D Tensile CIP Local 20x20x20'

```

```

/PREP7                                !Starts ANSYS
!Define geometry

K,1,0.01,0.01,0.01                    !Define keypoint 1 at (0.01,0.01,0.01) mm
K,2,0.01,0.00,0.01                    !Define keypoint 2 at (0.01,0.00,0.01) mm
K,3,0.00,0.00,0.01                    !Define keypoint 3 at (0.00,0.00,0.01) mm
K,4,0.00,0.01,0.01                    !Define keypoint 4 at (0.00,0.01,0.01) mm
A,1,2,3,4                              !Define area 1 from points 1-4
VOFFST,1,0.01                          !Define volume 1 by extruding area 1 10.0 microns

!Defines elastic material properties for material 1: 0.0% Coarse Grain Al-5083
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,70000                     !Defines Young's Modulus
MPDATA,PRXY,1,,0.33                   !Defines Poisson's Ratio

!Defines elastic material properties for material 2: 100% Coarse Grain Al-5083
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,2,,70000                     !Defines Young's Modulus
MPDATA,PRXY,2,,0.33                   !Defines Poisson's Ratio

!Defines plastic material properties for material 1: 0.0% Coarse Grain Al-5083
TB,MISO,1,1,78,0                       !Defines multilinear plastic properties
TBTEMP,0                               !for 0% Coarse Grain Al-5083 using
TBPT,,0.0080,560.0                     !Joshi's Voce Plasticity Model
TBPT,,0.0081,567.0
TBPT,,0.0082,574.0
TBPT,,0.0083,579.0
TBPT,,0.0084,581.5
TBPT,,0.0085,583.1
TBPT,,0.0086,584.6
TBPT,,0.0087,586.0
TBPT,,0.0088,587.5
TBPT,,0.0089,589.0
TBPT,,0.0090,590.4
TBPT,,0.0091,591.8
TBPT,,0.0092,593.2
TBPT,,0.0093,594.6
TBPT,,0.0094,596.0
TBPT,,0.0095,597.3
TBPT,,0.0096,598.6
TBPT,,0.0097,600.0
TBPT,,0.0098,601.3
TBPT,,0.0099,602.5
TBPT,,0.0100,603.8
TBPT,,0.0101,605.1
TBPT,,0.0102,606.3
TBPT,,0.0103,607.5
TBPT,,0.0104,608.8
TBPT,,0.0105,610.0
TBPT,,0.0106,611.1
TBPT,,0.0107,612.3
TBPT,,0.0108,613.5
TBPT,,0.0109,614.6
TBPT,,0.0110,615.7
TBPT,,0.0111,616.9
TBPT,,0.0112,618.0
TBPT,,0.0113,619.0
TBPT,,0.0114,620.1
TBPT,,0.0115,621.2
TBPT,,0.0116,622.2
TBPT,,0.0117,623.3
TBPT,,0.0118,624.3
TBPT,,0.0119,625.3
TBPT,,0.0120,626.3
TBPT,,0.0122,628.3

```

```

TBPT,,0.0124,630.2
TBPT,,0.0126,632.1
TBPT,,0.0128,633.9
TBPT,,0.0130,635.7
TBPT,,0.0132,637.5
TBPT,,0.0134,639.2
TBPT,,0.0136,640.8
TBPT,,0.0138,642.5
TBPT,,0.0140,644.0
TBPT,,0.0145,647.9
TBPT,,0.0150,651.4
TBPT,,0.0155,654.8
TBPT,,0.0160,658.0
TBPT,,0.0165,661.0
TBPT,,0.0170,663.9
TBPT,,0.0175,666.5
TBPT,,0.0180,669.0
TBPT,,0.0185,671.4
TBPT,,0.0190,673.6
TBPT,,0.0195,675.7
TBPT,,0.0200,677.7
TBPT,,0.0210,681.3
TBPT,,0.0220,684.6
TBPT,,0.0230,687.4
TBPT,,0.0240,690.0
TBPT,,0.0250,692.2
TBPT,,0.0260,694.2
TBPT,,0.0270,696.0
TBPT,,0.0280,697.5
TBPT,,0.0290,698.9
TBPT,,0.0300,700.2
TBPT,,0.0350,704.6
TBPT,,0.0400,707.0
TBPT,,0.0450,708.4
TBPT,,0.0500,709.1
TBPT,,0.1000,710.0
                                !Last stress/strain data point

!Define plastic material properties for material 2: 100% Coarse Grain Al-5083
TB,MISO,2,1,85,0                                !Defines multilinear plastic properties
TBTEMP,0                                          !for 100% Coarse Grain Al-5083 using
                                                !Joshi's Voce Plasticity Model
TBPT,,0.0040,280.0
TBPT,,0.0041,287.0
TBPT,,0.0042,293.0
TBPT,,0.0043,295.4
TBPT,,0.0044,295.7
TBPT,,0.0045,296.1
TBPT,,0.0046,296.5
TBPT,,0.0047,296.8
TBPT,,0.0048,297.2
TBPT,,0.0049,297.6
TBPT,,0.0050,297.9
TBPT,,0.0051,298.3
TBPT,,0.0052,298.7
TBPT,,0.0053,299.0
TBPT,,0.0054,299.4
TBPT,,0.0055,299.7
TBPT,,0.0056,300.1
TBPT,,0.0057,300.5
TBPT,,0.0058,300.8
TBPT,,0.0059,301.2
TBPT,,0.0060,301.5
TBPT,,0.0062,302.2
TBPT,,0.0064,302.9
TBPT,,0.0066,303.6
TBPT,,0.0068,304.3
TBPT,,0.0070,305.0
TBPT,,0.0072,305.7
TBPT,,0.0074,306.4

```

```

TBPT,,0.0076,307.0
TBPT,,0.0078,307.7
TBPT,,0.0080,308.4
TBPT,,0.0082,309.0
TBPT,,0.0084,309.7
TBPT,,0.0086,310.4
TBPT,,0.0088,311.0
TBPT,,0.0090,311.7
TBPT,,0.0092,312.3
TBPT,,0.0094,312.9
TBPT,,0.0096,313.6
TBPT,,0.0098,314.2
TBPT,,0.0100,314.6
TBPT,,0.0102,315.5
TBPT,,0.0104,316.1
TBPT,,0.0106,316.7
TBPT,,0.0108,317.3
TBPT,,0.0110,318.0
TBPT,,0.0112,318.6
TBPT,,0.0114,319.2
TBPT,,0.0116,319.8
TBPT,,0.0118,320.4
TBPT,,0.0120,321.0
TBPT,,0.0125,322.4
TBPT,,0.0130,323.9
TBPT,,0.0135,325.3
TBPT,,0.0140,326.7
TBPT,,0.0145,328.1
TBPT,,0.0150,329.5
TBPT,,0.0155,330.9
TBPT,,0.0160,332.2
TBPT,,0.0165,333.5
TBPT,,0.0170,334.8
TBPT,,0.0175,336.1
TBPT,,0.0180,337.4
TBPT,,0.0190,339.8
TBPT,,0.0200,342.2
TBPT,,0.0210,344.6
TBPT,,0.0220,346.8
TBPT,,0.0230,349.0
TBPT,,0.0240,351.2
TBPT,,0.0250,353.2
TBPT,,0.0275,358.2
TBPT,,0.0300,362.8
TBPT,,0.0325,367.1
TBPT,,0.0350,371.1
TBPT,,0.0375,374.8
TBPT,,0.0400,378.3
TBPT,,0.0425,381.5
TBPT,,0.0450,384.5
TBPT,,0.0475,387.3
TBPT,,0.0500,389.9
TBPT,,0.0550,394.5
TBPT,,0.0600,398.6
TBPT,,0.0700,405.2
TBPT,,0.0800,410.1
TBPT,,0.1000,416.6
!Last stress/strain data point

!Create dummy material 3 to change killed ufg elements to (material 1)
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,3,,70000
MPDATA,PRXY,3,,0.33
!Defines Young's Modulus
!Defines Poisson's Ratio

!Create dummy material 4 to change killed cg elements to (material 2)
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,4,,70000
!Defines Young's Modulus

```

```

MPDATA,PRXY,4,,0.33                                !Defines Poisson's Ratio

!Define element to mesh with
ET,2,SOLID186                                         !Defines SOLID186 element
LSEL,S,,,1,12                                         !Selects all ufg boundary lines (lines 1-12)
numb = 20                                             !Specifies 20 elements divisions per line
LESIZE,ALL,,,numb                                    !Applies 30 element divisions per line
ee=0.01/numb                                          !Computes the element size

CM,_Y,VOLU                                           !Makes a volume group named _Y
VSEL,,,,1                                           !Selects all volumes
CM,_Y1,VOLU                                          !Makes a volume group named _Y1
CHKMSH,'VOLU'                                       !Checks the volume for previous meshes
CMSEL,S,_Y                                           !Selects volume 1 to be meshed
VSWEEP,_Y1                                          !Meshes volume 1 using swept mapping
CMDELE,_Y                                           !Deletes volume group _Y
CMDELE,_Y1                                          !Deletes volume group _Y1
CMDELE,_Y2                                          !Deletes volume group _Y2

/PREP7                                               !Start the preprocessor
*DIM,XYZ,ARRAY,2,3                                  !Defines a XYZ array with 3 elements
(xloc,yloc,zloc)
*DIM,RL,ARRAY,2,2
*VFILL,XYZ(1,1),DATA,0.0
*VFILL,XYZ(1,2),DATA,0.0
*VFILL,XYZ(1,3),DATA,0.0
*VFILL,XYZ(2,1),DATA,0.005
*VFILL,XYZ(2,2),DATA,0.006
*VFILL,XYZ(2,3),DATA,0.006
Rin = ee
*VFILL,RL(1,1),DATA,0.003
*VFILL,RL(1,2),DATA,0.014
*VFILL,RL(2,1),DATA,0.0027
*VFILL,RL(2,2),DATA,0.006

*DO,ii,1,2,1
  /VIEW,1,1,,                                         !Changes the view to the global yz-plane
  /ANG,1
  /REP,FAST

  !Moves the origin of the working plane to XYZ(1),XYZ(2),XYZ(3) and aligns the new
  !z-direction with the global x-direction
  WPLANE,1,XYZ(ii,1),XYZ(ii,2),XYZ(ii,3),XYZ(ii,1),XYZ(ii,2),XYZ(ii,3)-0.1
  !Changes the coordinate system to cylindrical coordinates
  CSWPLA,11,1

  ESEL,NONE
  NSEL,NONE
  *DO,zz,(ee/2),RL(ii,2),ee
    rad = RL(ii,1) - ((RL(ii,1) - Rin)/RL(ii,2))*zz
    NSEL,A,LOC,Z,(zz - ee),(zz + ee)
    NSEL,R,LOC,X,0.0,rad
    ESLN,A,1
    EMODIF,ALL,MAT,2
  *ENDDO

  ESEL,NONE
  NSEL,NONE
  *DO,zz,-(ee/2),-RL(ii,2),-ee
    rad = RL(ii,1) + ((RL(ii,1) - Rin)/RL(ii,2))*zz
    NSEL,A,LOC,Z,(zz - ee),(zz + ee)
    NSEL,R,LOC,X,0.0,rad
    ESLN,A,1
    EMODIF,ALL,MAT,2
  *ENDDO
*ENDDO

```

```

ESEL,S,MAT,,2
*GET,CG,ELEM,,COUNT
ESEL,ALL
*GET,TOT,ELEM,,COUNT
cgfrac = CG/TOT

EPLOT
/VIEW,1,1,1,1          !Changes the view to an isometric view
/ANG,1
/REP,FAST

ALLSEL,ALL
CSYS,0
FINISH
SAVE,,,ALL
FINISH

/POST1
/PNUM,KP,0
/PNUM,LINE,0
/PNUM,AREA,0
/PNUM,VOLU,0
/PNUM,NODE,0
/PNUM,TABN,0
/PNUM,SVAL,0
/NUMBER,1
!*
/PNUM,MAT,1
/REPLOT
!*
esel,s,mat,,2
eplot

```

CIPped Local Solution

```

!Start 3D HIPped Tensile Analysis
!Requires geometry, elements, and material properties be input previously
/SOLU                      !Start solver
OUTRES,ERASE              !Erase output files
FINISH                    !Exit solver
*DIM,Fname,CHAR,30,1,1,, !Create a vector named Fname
*CREATE,ansuitmp          !Create an input macro file
*VREAD,Fname,'Fname','txt',' ',IJK,30,1,1,, !Read from Fname.txt to Fname vector
(A)                      !Read input in character format
*END                      !Close the macro file
/INPUT,ansuitmp           !Switch the input file for following commands
FINISH                    !Finish the file input

!Apply boundary conditions and input load file
/PREP7                    !Start the preprocessor
!_____!
!Apply symmetry to x, y, and z boundaries at 0
NSEL,S,LOC,X,0.0          !Select nodes at x = 0.0
DSYM,SYMM,X              !Apply symmetry condition
NSEL,S,LOC,Y,0.0          !Select nodes at y = 0.0
DSYM,SYMM,Y              !Apply symmetry condition
NSEL,S,LOC,Z,0.0          !Select nodes at z = 0.0
DSYM,SYMM,Z              !Apply symmetry condition
ALLSEL,ALL               !Reselect all nodes
FINISH                    !Exit preprocessor

!Read input file to 2D array
*DIM,loads,ARRAY,30,2,1
*CREATE,ansuitmp
*VREAD,loads,'loads','txt',,JIK,2,30,1
(F9.7,2X,F10.7)

```



```

*END
/INPUT,ansuitmp
FINISH

*DO,ii,1,2,1
*DO,jj,1,30,1
loads(jj,ii,1)=loads(jj,ii,1)*(6.25/3.00)
*ENDDO
*ENDDO
!_____!

!Iterate through the loads from the global model
!_____!
*DO,q,21,30,1
FINISH                                !Exit solver
/FILNAME,Fname(q,1,1),1              !Create output filename

/PREP7                                !Start the preprocessor
ALLSEL,ALL                            !Select all nodes and elements

!Apply displacements from local model
NSEL,S,LOC,X,0.01                      !Select nodes at x = 0.01
DDELE,ALL,UX                           !Delete existing displacements
D,ALL,UX,loads(q,1,1)                  !Displace selected nodes
NSEL,S,LOC,Y,0.01                      !Select nodes at y = 0.01
DDELE,ALL,UY                           !Delete existing displacements
D,ALL,UY,loads(q,2,1)                  !Displace selected nodes
NSEL,S,LOC,Z,0.01                      !Select nodes at z = 0.01
DDELE,ALL,UZ                           !Delete existing displacements
D,ALL,UZ,loads(q,2,1)                  !Displace selected nodes
ALLSEL,ALL                             !Reselect all nodes
FINISH                                 !Exit the preprocessor

!Solve the load step
/SOLU                                  !Start the solver
NSUBST,100                             !Specify 100 substeps
SOLVE                                  !Solve the load step
FINISH                                 !Exit the solver

/POST1                                 !Start the Postprocessor
ALLSEL,ALL                             !Selects all elements and nodes
ESEL,S,MAT,,2                          !Picks all CG elements
ETABLE,cgs,S,EQV                       !Create element table with equivalent stresses
ESEL,R,ETAB,cgs,390.0                  !Pick elements that have exceeded 390.0 stress

/SOLU                                  !Start the solver
EKILL,ALL                              !Kills selected elements
FINISH                                 !Exit the solver

/PREP7                                 !Start the preprocessor
EMODIF,ALL,MAT,4                       !Change killed CG elements to material 4
FINISH                                 !Exit the preprocessor

/POST1                                 !Start the postprocessor
ALLSEL,ALL                             !Selects all elements and nodes
ESEL,S,MAT,,1                          !Picks all UFG elements
ETABLE,ufgs,S,EQV                     !Create element table with the equivalent stresses
ESEL,R,ETAB,ufgs,690.0                !Pick elements that have exceeded 690.0 stress

/SOLU                                  !Start the solver
EKILL,ALL                              !Kills selected elements
FINISH                                 !Exit the solver

/PREP7                                 !Start the preprocessor
EMODIF,ALL,MAT,3                       !Changes killed UFG elements to material 3
FINISH                                 !Exit the preprocessor

SAVE,,,ALL                            !Save the current solution to b,c,d,...,txt

```



```

TBPT,,0.0073,397.251
TBPT,,0.0074,397.989
TBPT,,0.0075,398.716
TBPT,,0.0076,399.433
TBPT,,0.0077,400.140
TBPT,,0.0078,400.837
TBPT,,0.0079,401.524
TBPT,,0.0080,402.202
TBPT,,0.0082,403.529      !30 data points
TBPT,,0.0084,404.219
TBPT,,0.0086,406.074
TBPT,,0.0088,407.294
TBPT,,0.0090,408.480
TBPT,,0.0092,409.633
TBPT,,0.0094,410.754
TBPT,,0.0096,411.844
TBPT,,0.0098,412.904
TBPT,,0.0100,413.934
TBPT,,0.0105,416.386      !40 data points
TBPT,,0.0110,418.672
TBPT,,0.0115,420.803
TBPT,,0.0120,422.789
TBPT,,0.0125,424.641
TBPT,,0.0130,426.366
TBPT,,0.0135,427.974
TBPT,,0.0140,429.473
TBPT,,0.0145,430.871
TBPT,,0.0150,432.173
TBPT,,0.0155,433.387      !50 data points
TBPT,,0.0160,434.519
TBPT,,0.0165,435.573
TBPT,,0.0170,436.556
TBPT,,0.0175,437.473
TBPT,,0.0180,438.327
TBPT,,0.0190,439.865
TBPT,,0.0200,441.201
TBPT,,0.0210,442.362
TBPT,,0.0220,443.371
TBPT,,0.0230,444.247      !60 data points
TBPT,,0.0240,445.009
TBPT,,0.0250,445.670
TBPT,,0.0300,447.882
TBPT,,0.0350,448.977
TBPT,,0.0400,449.519
TBPT,,0.0450,449.787
TBPT,,0.0500,449.918
TBPT,,0.0600,450.018
TBPT,,0.0700,450.042
TBPT,,0.0800,450.048      !70 data points
TBPT,,0.0900,450.050
TBPT,,0.1000,450.050
TBPT,,0.1100,450.050      !Last stress/strain data point

ET,2,SOLID186              !Defines element to mesh with

LSEL,S,LOC,X,0.0
LSEL,A,LINE,,18,23
LESIZE,ALL,,5
LSEL,ALL
LESIZE,01,0.1
LESIZE,09,0.1
LESIZE,07,0.1
LESIZE,15,0.1
LESIZE,02,,18
LESIZE,10,,18
LESIZE,06,,18
LESIZE,14,,18
LESIZE,03,,7

```

CM,_Y,VOLU	!Makes a volume group named _Y
VSEL,,,1	!Selects volume 1
CM,_Y1,VOLU	!Makes a volume group named _Y1
CHKMSH,'VOLU'	!Checks the volume for previous meshes
CMSEL,S,_Y	!Selects volume 1 to be meshed
VSWEEP,_Y1	!Meshes volume 1 using swept mapping
CMDELE,_Y	!Deletes volume group _Y
CMDELE,_Y1	!Deletes volume group _Y1
CMDELE,_Y2	!Deletes volume group _Y2

ALLSEL,ALL
FINISH
SAVE,,,,ALL

```

!Start 3D HIPped Tensile Analysis
!Requires geometry, elements, and material properties be input previously
/SOLU                                !Start Solver
OUTRES,ERASE                         !Erase output files
FINISH                               !Exit Solver

*DIM,Fname,CHAR,30,1,1,,,           !Create a vector name Fname
*CREATE,ansuitmp                     !Create an input macro file
*VREAD,Fname,'Fname','txt',' ',IJK,30,1,1,, !Read from Fname.txt to Fname vector
(A)                                  !Read input in character format
*END                                  !Close the macro file
/INPUT,ansuitmp                     !Switch the input file for following commands
FINISH                               !Finish the file input

!Apply boundary conditions
/PREP7                                !Start the preprocessor
!_____!
!Apply symmetry to x y and z boundaries at 0
NSEL,S,LOC,X,0.0                     !Select nodes at x = 0.00
DSYM,SYMM,X                          !Apply symmetry conditions
NSEL,S,LOC,Y,0.0                     !Select nodes at y = 0.00
DSYM,SYMM,Y                          !Apply symmetry conditions
NSEL,S,LOC,Z,0.0                     !Select nodes at z = 0.00
DSYM,SYMM,Z                          !Apply symmetry conditions
ALLSEL,ALL                           !Reselect all nodes

!Solve using incremental steps starting at 80% of the yield strength and ending at the
!ultimate tensile strength
yield_str = 380.0                    !Define the yield strength
ulten_str = 450.0                    !Define the UTS
gage_len = 3.00                      !Define the gage length
x_disp_ini = 380.0*gage_len/72500.0 !Define the starting displacement
x_disp_fin = 450.0*gage_len/72500.0 !Define the final displacement
num_steps = 30.0                     !Define the number of steps
step_size = (x_disp_fin - x_disp_ini)/num_steps !Define the step size

*DO,q,1,2*num_steps/3,1
FINISH                                !Exit the solver
/FILNAME,Fname(q,1,1),1              !Create output filename
/PREP7                                !Start the preprocessor
ASEL,S,AREA,,4                       !Select the pinhole surface
NSLA,,1                               !Select the nodes associated with the
pinhole
NSEL,R,LOC,X,5.0,6.25                !Select the upper pinhole surface

```

```

DDELE,ALL,UX                                !Delete existing displacements
x_disp = x_disp_ini + q*step_size           !Define current load value
D,ALL,UX,x_disp                             !Displace the pinhole nodes
ALLSEL,ALL                                  !Reselect everything
FINISH                                       !Exit the preprocessor

!Start load steps
/SOLU                                       !Start the solver
NSUBST,100                                !Specify 100 substeps
SOLVE                                       !Solve the load step
FINISH                                       !Exit the solver

ALLSEL,ALL                                  !Reselect all nodes
SAVE,,,ALL                                  !Save all database information
FINISH                                       !Exit save

*ENDDO

```

HIPped Local Model

```

!Tensile model containing geometry, material, and element information for
!HIPped material - All dimensions are in millimeters
/FILNAME,model,0
/CWD,'C:\ANSYS\3D Tensile HIP Local 20x20x20'

/PREP7                                     !Starts ANSYS
!Define geometry

K,1,0.01,0.01,0.01                         !Define keypoint 1 at (0.01,0.01,0.01) mm
K,2,0.01,0.00,0.01                         !Define keypoint 2 at (0.01,0.00,0.01) mm
K,3,0.00,0.00,0.01                         !Define keypoint 3 at (0.00,0.00,0.01) mm
K,4,0.00,0.01,0.01                         !Define keypoint 4 at (0.00,0.01,0.01) mm
A,1,2,3,4                                   !Define area 1 from points 1-4
VOFFST,1,0.01                              !Define volume 1 by extruding area 1 10.0 microns

!Defines elastic material properties for material 1: 0.0% Coarse Grain Al-5083
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,72500                          !Defines Young's Modulus
MPDATA,PRXY,1,,0.33                         !Defines Poisson's Ratio

!Defines elastic material properties for material 2: 100% Coarse Grain Al-5083
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,2,,72500                          !Defines Young's Modulus
MPDATA,PRXY,2,,0.33                         !Defines Poisson's Ratio

!Defines plastic material properties for material 1: 0.0% Coarse Grain Al-5083
TB,MISO,1,1,93,0                           !Defines multilinear plastic properties
TBTEMP,0                                    !for 0% Coarse Grain Al-5083 using
TBPT,,0.0060,435.000                        !Joshi's Voce Plasticity Model
TBPT,,0.0061,441.000
TBPT,,0.0062,441.485
TBPT,,0.0063,441.965
TBPT,,0.0064,442.438
TBPT,,0.0065,442.907
TBPT,,0.0066,443.369
TBPT,,0.0067,443.826
TBPT,,0.0068,444.278
TBPT,,0.0069,444.725                        !10 data points
TBPT,,0.0070,445.166
TBPT,,0.0071,445.602
TBPT,,0.0072,446.032
TBPT,,0.0073,446.458
TBPT,,0.0074,446.879
TBPT,,0.0075,447.294

```

```

TBPT,,0.0076,447.705
TBPT,,0.0077,448.111
TBPT,,0.0078,448.512
TBPT,,0.0079,448.908
TBPT,,0.0080,449.230
TBPT,,0.0081,449.687
TBPT,,0.0082,450.069
TBPT,,0.0083,450.447
TBPT,,0.0084,450.820
TBPT,,0.0085,451.189
TBPT,,0.0086,451.554
TBPT,,0.0087,451.914
TBPT,,0.0088,452.270
TBPT,,0.0089,452.622
TBPT,,0.0090,452.970
TBPT,,0.0091,453.313
TBPT,,0.0092,453.653
TBPT,,0.0093,453.988
TBPT,,0.0094,454.320
TBPT,,0.0095,454.647
TBPT,,0.0096,454.971
TBPT,,0.0097,455.291
TBPT,,0.0098,455.607
TBPT,,0.0099,455.919
TBPT,,0.0100,456.228
TBPT,,0.0102,456.834
TBPT,,0.0104,457.427
TBPT,,0.0106,458.005
TBPT,,0.0108,458.569
TBPT,,0.0110,459.121
TBPT,,0.0112,459.659
TBPT,,0.0114,460.185
TBPT,,0.0116,460.698
TBPT,,0.0118,461.199
TBPT,,0.0120,461.688
TBPT,,0.0122,462.166
TBPT,,0.0124,462.633
TBPT,,0.0126,463.089
TBPT,,0.0128,463.534
TBPT,,0.0130,463.968
TBPT,,0.0132,464.392
TBPT,,0.0134,464.807
TBPT,,0.0136,465.211
TBPT,,0.0138,465.606
TBPT,,0.0140,465.992
TBPT,,0.0145,466.917
TBPT,,0.0150,467.789
TBPT,,0.0155,468.610
TBPT,,0.0160,469.384
TBPT,,0.0165,470.113
TBPT,,0.0170,470.800
TBPT,,0.0175,471.447
TBPT,,0.0180,472.057
TBPT,,0.0185,472.631
TBPT,,0.0190,473.173
TBPT,,0.0195,473.683
TBPT,,0.0200,474.163
TBPT,,0.0210,475.043
TBPT,,0.0220,475.824
TBPT,,0.0230,476.517
TBPT,,0.0240,477.132
TBPT,,0.0250,477.679
TBPT,,0.0260,478.164
TBPT,,0.0270,478.594
TBPT,,0.0280,479.976
TBPT,,0.0290,479.316
TBPT,,0.0300,479.617
TBPT,,0.0350,480.686

```

!20 data points

!30 data points

!40 data points

!50 data points

!60 data points

!70 data points

!80 data points

```

TBPT,,0.0400,481.275
TBPT,,0.0450,481.600
TBPT,,0.0500,481.780
TBPT,,0.0600,481.933
TBPT,,0.0700,481.980
TBPT,,0.0800,481.994      !90 data points
TBPT,,0.0900,481.998
TBPT,,0.1000,481.999
TBPT,,0.1070,482.000      !Last stress/strain data point

!Define plastic material properties for material 2: 100% Coarse Grain Al-5083
TB,MISO,2,1,33,0          !Defines multilinear plastic properties
TBTEMP,0                  !for 100% Coarse Grain Al-5083 using
                           !Joshi's Voce Plasticity Model
TBPT,,0.0039,282.750
TBPT,,0.0040,286.000
TBPT,,0.0045,286.683
TBPT,,0.0050,287.332
TBPT,,0.0055,287.950
TBPT,,0.0060,288.538
TBPT,,0.0065,289.067
TBPT,,0.0070,289.628
TBPT,,0.0075,290.134
TBPT,,0.0080,290.616      !10 data points
TBPT,,0.0085,291.073
TBPT,,0.0090,291.509
TBPT,,0.0095,291.923
TBPT,,0.0100,292.316
TBPT,,0.0110,293.048
TBPT,,0.0120,293.709
TBPT,,0.0130,294.308
TBPT,,0.0140,294.850
TBPT,,0.0150,295.340
TBPT,,0.0160,295.783      !20 data points
TBPT,,0.0170,296.185
TBPT,,0.0180,296.548
TBPT,,0.0190,296.876
TBPT,,0.0200,297.173
TBPT,,0.0300,298.960
TBPT,,0.0400,299.617
TBPT,,0.0500,299.859
TBPT,,0.0600,299.948
TBPT,,0.0700,299.981
TBPT,,0.0800,299.993      !30 data points
TBPT,,0.0900,299.997
TBPT,,0.1000,299.999
TBPT,,0.1080,300.000      !Last stress/strain data point

!Create dummy material 3 to change killed ufg elements to (material 1)
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,3,,72500        !Defines Young's Modulus
MPDATA,PRXY,3,,0.33       !Defines Poisson's Ratio

!Create dummy material 4 to change killed cg elements to (material 2)
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,4,,72500        !Defines Young's Modulus
MPDATA,PRXY,4,,0.33       !Defines Poisson's Ratio

!Define element to mesh with
ET,2,SOLID186              !Defines SOLID186 element
LSEL,S,,,1,12              !Selects all ufg boundary lines (lines 1-12)
numb = 20                  !Specifies 30 elements divisions per line
LESIZE,ALL,,,numb          !Applies 30 element divisions per line
ee=0.01/numb               !Computes the element size

CM,_Y,VOLU                 !Makes a volume group named _Y

```

```

VSEL,,,1                                !Selects all volumes
CM,_Y1,VOLU                             !Makes a volume group named _Y1
CHKMSH,'VOLU'                           !Checks the volume for previous meshes
CMSEL,S,_Y                               !Selects volume 1 to be meshed
VSWEPT,_Y1                              !Meshes volume 1 using swept mapping
CMDELE,_Y                                !Deletes volume group _Y
CMDELE,_Y1                              !Deletes volume group _Y1
CMDELE,_Y2                              !Deletes volume group _Y2

/PREP7                                  !Start the preprocessor
*DIM,XYZ,ARRAY,2,3                      !Defines a XYZ array with 3 elements
(xloc,yloc,zloc)
*DIM,RL,ARRAY,2,2
*VFILL,XYZ(1,1),DATA,0.0
*VFILL,XYZ(1,2),DATA,0.0
*VFILL,XYZ(1,3),DATA,0.0
*VFILL,XYZ(2,1),DATA,0.005
*VFILL,XYZ(2,2),DATA,0.006
*VFILL,XYZ(2,3),DATA,0.006
Rin = ee
*VFILL,RL(1,1),DATA,0.003
*VFILL,RL(1,2),DATA,0.014
*VFILL,RL(2,1),DATA,0.0027
*VFILL,RL(2,2),DATA,0.006

*DO,ii,1,2,1
  /VIEW,1,1,,                          !Changes the view to the global yz-plane
  /ANG,1
  /REP,FAST

  !Moves the origin of the working plane to XYZ(1),XYZ(2),XYZ(3) and aligns the new
  !z-direction with the global x-direction
  WPLANE,1,XYZ(ii,1),XYZ(ii,2),XYZ(ii,3),XYZ(ii,1),XYZ(ii,2),XYZ(ii,3)-0.1
  !Changes the coordinate system to cylindrical coordinates
  CSWPLA,11,1

  ESEL,NONE
  NSEL,NONE
  *DO,zz,(ee/2),RL(ii,2),ee
    rad = RL(ii,1) - ((RL(ii,1) - Rin)/RL(ii,2))*zz
    NSEL,A,LOC,Z,(zz - ee),(zz + ee)
    NSEL,R,LOC,X,0.0,rad
    ESLN,A,1
    EMODIF,ALL,MAT,2
  *ENDDO

  ESEL,NONE
  NSEL,NONE
  *DO,zz,-(ee/2),-RL(ii,2),-ee
    rad = RL(ii,1) + ((RL(ii,1) - Rin)/RL(ii,2))*zz
    NSEL,A,LOC,Z,(zz - ee),(zz + ee)
    NSEL,R,LOC,X,0.0,rad
    ESLN,A,1
    EMODIF,ALL,MAT,2
  *ENDDO
*ENDDO

ESEL,S,MAT,,2
*GET,CG,ELEM,,COUNT
ESEL,ALL
*GET,TOT,ELEM,,COUNT
cgfrac = CG/TOT

EPLOT
/VIEW,1,1,1,1                          !Changes the view to an isometric view
/ANG,1
/REP,FAST

```



```

ALLSEL,ALL
CSYS,0
FINISH
SAVE,,,ALL
FINISH

/POST1
/PNUM,KP,0
/PNUM,LINE,0
/PNUM,AREA,0
/PNUM,VOLU,0
/PNUM,NODE,0
/PNUM,TABN,0
/PNUM,SVAL,0
/NUMBER,1
!*
/PNUM,MAT,1
/REPLOT
!*
esel,s,mat,,2
epplot

```

HIPped Local Solution

```

!Start 3D HIPped Tensile Analysis
!Requires geometry, elements, and material properties be input previously
/SOLU                                !Start solver
OUTRES,ERASE                         !Erase output files
FINISH                               !Exit solver
*DIM,Fname,CHAR,30,1,1,,            !Create a vector named Fname
*CREATE,ansuitmp                     !Create an input macro file
*VREAD,Fname,'Fname','txt',' ',,IJK,30,1,1,, !Read from Fname.txt to Fname vector
(A)                                  !Read input in character format
*END                                 !Close the macro file
/INPUT,ansuitmp                     !Switch the input file for following commands
FINISH                              !Finish the file input

!Apply boundary conditions and input load file
/PREP7                               !Start the preprocessor
!_____!
!Apply symmetry to x, y, and z boundaries at 0
NSEL,S,LOC,X,0.0                    !Select nodes at x = 0.0
DSYM,SYMM,X                         !Apply symmetry condition
NSEL,S,LOC,Y,0.0                    !Select nodes at y = 0.0
DSYM,SYMM,Y                         !Apply symmetry condition
NSEL,S,LOC,Z,0.0                    !Select nodes at z = 0.0
DSYM,SYMM,Z                         !Apply symmetry condition
ALLSEL,ALL                          !Reselect all nodes
FINISH                              !Exit preprocessor

!Read input file to 2D array
*DIM,loads,ARRAY,20,2,1
*CREATE,ansuitmp
*VREAD,loads,'loads','txt',,JIK,2,20,1
(F9.7,2X,F11.8)
*END
/INPUT,ansuitmp
FINISH

*DO,ii,1,2,1
*DO,jj,1,20,1
loads(jj,ii,1)=loads(jj,ii,1)*(6.25/3.00)
*ENDDO
*ENDDO
!_____!

```

```

!Iterate through the loads from the global model
!
*DO,q,1,10,1
FINISH                                !Exit solver
/FILNAME,Fname(q,1,1),1              !Create output filename

/PREP7                                !Start the preprocessor
ALLSEL,ALL                            !Select all nodes and elements

!Apply displacements from local model
NSEL,S,LOC,X,0.01                     !Select nodes at x = 0.01
DDELE,ALL,UX                          !Delete existing displacements
D,ALL,UX,loads(q,1,1)                 !Displace selected nodes
NSEL,S,LOC,Y,0.01                     !Select nodes at y = 0.01
DDELE,ALL,UY                          !Delete existing displacements
D,ALL,UY,loads(q,2,1)                 !Displace selected nodes
NSEL,S,LOC,Z,0.01                     !Select nodes at z = 0.01
DDELE,ALL,UZ                          !Delete existing displacements
D,ALL,UZ,loads(q,2,1)                 !Displace selected nodes
ALLSEL,ALL                            !Reselect all nodes
FINISH                                !Exit the preprocessor

!Solve the load step
/SOLU                                  !Start the solver
NSUBST,100                            !Specify 100 substeps
SOLVE                                  !Solve the load step
FINISH                                !Exit the solver

/POST1                                 !Start the Postprocessor
ALLSEL,ALL                            !Selects all elements and nodes
ESEL,S,MAT,,2                         !Picks all CG elements
ETABLE,cgs,S,EQV                      !Create element table with equivalent stresses
ESEL,R,ETAB,cgs,290.0                 !Pick elements that have exceeded 390.0 stress

/SOLU                                  !Start the solver
EKILL,ALL                             !Kills selected elements
FINISH                                !Exit the solver

/PREP7                                 !Start the preprocessor
EMODIF,ALL,MAT,4                      !Change killed CG elements to material 4
FINISH                                !Exit the preprocessor

/POST1                                 !Start the postprocessor
ALLSEL,ALL                            !Selects all elements and nodes
ESEL,S,MAT,,1                         !Picks all UFG elements
ETABLE,ufgs,S,EQV                    !Create element table with the equivalent stresses
ESEL,R,ETAB,ufgs,440.0               !Pick elements that have exceeded 690.0 stress

/SOLU                                  !Start the solver
EKILL,ALL                             !Kills selected elements
FINISH                                !Exit the solver

/PREP7                                 !Start the preprocessor
EMODIF,ALL,MAT,3                      !Changes killed UFG elements to material 3
FINISH                                !Exit the preprocessor

SAVE,,,ALL                            !Save the current solution to b,c,d,...,txt
FINISH                                !Exit the save
*ENDDO

```

Cyclic 2D Input Files

Global Model

```
!Builds the 2D global model
```

```

!All dimensions are in millimeters
/FILNAME,model,0
/CWD,'C:\ANSYS\2D Fatigue Global'
/PREP7                                !Starts ANSYS

!Define geometry!
K,1,0.00,0.00,0.00                    !Define keypoint 01 at (0.00,0.00,0.00)
K,2,3.75,0.00,0.00                    !Define keypoint 02 at (3.75,0.00,0.00)
K,3,5.00,1.25,0.00                    !Define keypoint 03 at (5.00,1.25,0.00)
K,4,6.25,0.00,0.00                    !Define keypoint 04 at (6.25,0.00,0.00)
K,5,8.00,0.00,0.00                    !Define keypoint 05 at (8.00,0.00,0.00)
K,6,8.00,3.00,0.00                    !Define keypoint 06 at (8.00,3.00,0.00)
K,7,3.00,3.00,0.00                    !Define keypoint 07 at (3.00,3.00,0.00)
K,8,3.00,0.50,0.00                    !Define keypoint 08 at (3.00,0.50,0.00)
K,9,0.00,0.50,0.00                    !Define keypoint 09 at (0.00,0.50,0.00)

L,1,2                                !Define line 01 between points 1 and 2
LARC,2,4,3,1.25                       !Define line 02 as arc with rad 1.25 between points 2,3,4
L,4,5                                !Define line 03 between points 4 and 5
L,5,6                                !Define line 04 between points 5 and 6
L,6,7                                !Define line 05 between points 6 and 7
LFILLT,4,5,1.25                       !Define line 06 (fillet) with rad 1.25 between lines 4 & 5
LCOMB,4,6                             !Define line 04 by combining lines 4 and 6 (06 is deleted)
L,7,8                                !Define line 06 between points 7 and 8
L,8,9                                !Define line 07 between points 8 and 9
LFILLT,6,7,1.25                       !Define line 08 (fillet) with rad 1.25 between lines 7 & 8
LCOMB,6,8                             !Define line 06 by combining lines 6 and 8 (08 is deleted)
L,9,1                                !Define line 08 between points 9 and 1

AL,1,2,3,4,5,6,7,8                   !Creates area from lines 1, 2, 3, 4, 5, 6, 7, and 8
!VOFFST,1,0.5                         !Creates volume from area 1 by offsetting 0.5 mm in +z-
direction

!Define global model as 15% cg content from Joshi's model
MPTEMP,,,,,,,,                       !Defines elastic material properties
MPTEMP,1,0
MPDATA,EX,1,,72500                   !Defines Young's Modulus
MPDATA,PRXY,1,,0.33                  !Defines Poisson's Ratio

TB,MISO,1,1,73,0                     !Defines multilinear plastic properties
TBTEMP,0
TBPT,,0.0052,377.000
TBPT,,0.0053,380.100
TBPT,,0.0054,381.077
TBPT,,0.0055,382.040
TBPT,,0.0056,382.990
TBPT,,0.0057,383.927
TBPT,,0.0058,384.850
TBPT,,0.0059,385.761
TBPT,,0.0060,386.659
TBPT,,0.0061,387.544                 !10 data points
TBPT,,0.0062,388.417
TBPT,,0.0063,389.278
TBPT,,0.0064,390.126
TBPT,,0.0065,390.963
TBPT,,0.0066,391.789
TBPT,,0.0067,392.602
TBPT,,0.0068,393.405
TBPT,,0.0069,394.196
TBPT,,0.0070,394.976
TBPT,,0.0071,395.745                 !20 data points
TBPT,,0.0072,396.503
TBPT,,0.0073,397.251
TBPT,,0.0074,397.989
TBPT,,0.0075,398.716
TBPT,,0.0076,399.433
TBPT,,0.0077,400.140
TBPT,,0.0078,400.837

```

```

TBPT,,0.0079,401.524
TBPT,,0.0080,402.202
TBPT,,0.0082,403.529      !30 data points
TBPT,,0.0084,404.219
TBPT,,0.0086,406.074
TBPT,,0.0088,407.294
TBPT,,0.0090,408.480
TBPT,,0.0092,409.633
TBPT,,0.0094,410.754
TBPT,,0.0096,411.844
TBPT,,0.0098,412.904
TBPT,,0.0100,413.934
TBPT,,0.0105,416.386      !40 data points
TBPT,,0.0110,418.672
TBPT,,0.0115,420.803
TBPT,,0.0120,422.789
TBPT,,0.0125,424.641
TBPT,,0.0130,426.366
TBPT,,0.0135,427.974
TBPT,,0.0140,429.473
TBPT,,0.0145,430.871
TBPT,,0.0150,432.173
TBPT,,0.0155,433.387      !50 data points
TBPT,,0.0160,434.519
TBPT,,0.0165,435.573
TBPT,,0.0170,436.556
TBPT,,0.0175,437.473
TBPT,,0.0180,438.327
TBPT,,0.0190,439.865
TBPT,,0.0200,441.201
TBPT,,0.0210,442.362
TBPT,,0.0220,443.371
TBPT,,0.0230,444.247      !60 data points
TBPT,,0.0240,445.009
TBPT,,0.0250,445.670
TBPT,,0.0300,447.882
TBPT,,0.0350,448.977
TBPT,,0.0400,449.519
TBPT,,0.0450,449.787
TBPT,,0.0500,449.918
TBPT,,0.0600,450.018
TBPT,,0.0700,450.042
TBPT,,0.0800,450.048      !70 data points
TBPT,,0.0900,450.050
TBPT,,0.1000,450.050
TBPT,,0.1100,450.050      !Last stress/strain data point

ET,1,PLANE183              !Defines element to mesh with
KEYOPT,1,1,0               !Indicates a full integration
KEYOPT,1,3,0               !Indicates plane strain
KEYOPT,1,6,0               !Indicates pure displacement
KEYOPT,1,10,0              !Indicates no user defined initial stress

LESIZE,08,,50
LESIZE,01,0.01
LESIZE,07,0.01
LESIZE,02,,60
LESIZE,06,,60
LESIZE,03,,7
LESIZE,04,,12
LESIZE,05,,15

!Mesh the model
MSHKEY,0
CM,_Y,AREA
ASEL,,,1
CM,_Y1,AREA
CHKMSH,'AREA'

```

```

CMSEL,S,_Y
AMESH,_Y1
CMDELE,_Y
CMDELE,_Y1
CMDELE,_Y2

ALLSEL,ALL                !Reselect all nodes
FINISH                    !Exit the preprocessor
SAVE,,,ALL                !Save the model and the constraints

```

Global Solution

```

!Start 2D global fatigue analysis
!Requires geometry, elements and material properties be input previously

!Apply boundary conditions and determine loads
/FILNAME,constraints,0    !Specify filename
/PREP7                    !Start preprocessor
!_____!

!Apply symmetry to x, y and z boundaries at 0
NSEL,S,LOC,x,0.0          !Select all nodes on yz-plane at x = 0.0
DSYM,SYMM,X               !Apply symmetry conditions to selected nodes
NSEL,S,LOC,y,0.0          !Select all nodes on xz-plane at y = 0.0
DSYM,SYMM,Y               !Apply symmetry conditions to selected nodes

!Determine appropriate displacement from plastic strain amplitude
strn_amp = 0.001          !Sets the constant plastic strain amplitude
gage_len = 6.25           !Sets the gage length
x_disp = gage_len*(450.0/72500.0 + strn_amp) !Determines the required displacement
ALLSEL,ALL                !Reselect all nodes
FINISH                    !Exit preprocessor
SAVE,,,ALL                !Save all database information
!_____!

!Start Tension Portion of First Cycle
/FILNAME,tension_cycle_01,1 !Specify Filename
/PREP7                    !Start preprocessor
!_____!

!Apply tension displacement
!NSEL,S,NODE,,10000        !Selects nodes 10000 - 10005
LSEL,S,LINE,,2            !Select Line 2
NSLL,,1                   !Select nodes associated with line 2
DDELE,ALL,UX              !Delete all constraints from selected nodes
NSEL,R,LOC,X,5.0,6.25     !Select only the nodes loaded in tension
D,ALL,UX,x_disp           !Displace the selected nodes
ALLSEL,ALL                !Reselect all nodes
FINISH                    !Exit the preprocessor

!Start tensile solution
/SOLU                     !Start the solver
SOLVE                     !Solve the load step
FINISH                    !Exit the solver

!Determine the displacement on the element at (0,0,0)
/POST1                    !Start the post-processor
ALLSEL,ALL                !Reselect all nodes

FINISH                    !Exit the post-processor
SAVE,,,ALL                !Save all database information
FINISH
!_____!

!Start Compression Portion of First Cycle
/FILENAME,compress_cycle01,1 !Specify Filename
/PREP7                    !Start Preprocessor
!_____!

```

```

!Apply Compression displacement
LSEL,S,LINE,,2          !Select Line 2
NSLL,,1                !Select nodes associated with line 2
DDELE,ALL,UX            !Delete all constraints from selected nodes
NSEL,R,LOC,X,3.75,5.0   !Select only the nodes loaded in compression
D,ALL,UX,-x_disp        !Displace the selected nodes
ALLSEL,ALL              !Reselect all nodes
FINISH                  !Exit the preprocessor

!Start Compressive solution
/SOLU                   !Start the solver
SOLVE                   !Solve the load step
FINISH                  !Exit the solver

SAVE,,,ALL              !Save all database information
FINISH

!_____!

```

Local Model

```

!Fatigue model containing geometry, material, and element information for
!fatigue model - All dimensions are in millimeters
/FILNAME,model,0
/CWD,'C:\ANSYS\2D Fatigue Local 10CG 200x200'

/PREP7                  !Starts preprocessor
!Define geometry
K,1,0.00,0.00,0.0       !Define keypoint 1 at (0.00,0.00,0.0) mm
K,2,0.01,0.00,0.0       !Define keypoint 2 at (0.01,0.00,0.0) mm
K,3,0.01,0.01,0.0       !Define keypoint 3 at (0.01,0.01,0.0) mm
K,4,0.00,0.01,0.0       !Define keypoint 4 at (0.00,0.01,0.0) mm
A,1,2,3,4                !Define area 1 from points 1-4

!Defines elastic material properties for material 1: 0.0% Coarse Grain Al-5083
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,72500       !Defines Young's Modulus
MPDATA,PRXY,1,,0.33      !Defines Poisson's Ratio

!Defines elastic material properties for material 2: 100% Coarse Grain Al-5083
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,2,,72500       !Defines Young's Modulus
MPDATA,PRXY,2,,0.33      !Defines Poisson's Ratio

!Defines plastic material properties for material 1: 0.0% Coarse Grain Al-5083
TB,MISO,1,1,93,0         !Defines multilinear plastic properties
TBTEMP,0                 !for 0% Coarse Grain Al-5083 using
TBPT,,0.0060,435.000     !Joshi's Voce Plasticity Model
TBPT,,0.0061,441.000
TBPT,,0.0062,441.485
TBPT,,0.0063,441.965
TBPT,,0.0064,442.438
TBPT,,0.0065,442.907
TBPT,,0.0066,443.369
TBPT,,0.0067,443.826
TBPT,,0.0068,444.278
TBPT,,0.0069,444.725     !10 data points
TBPT,,0.0070,445.166
TBPT,,0.0071,445.602
TBPT,,0.0072,446.032
TBPT,,0.0073,446.458
TBPT,,0.0074,446.879
TBPT,,0.0075,447.294
TBPT,,0.0076,447.705
TBPT,,0.0077,448.111

```

```

TBPT,,0.0078,448.512
TBPT,,0.0079,448.908
TBPT,,0.0080,449.230
TBPT,,0.0081,449.687
TBPT,,0.0082,450.069
TBPT,,0.0083,450.447
TBPT,,0.0084,450.820
TBPT,,0.0085,451.189
TBPT,,0.0086,451.554
TBPT,,0.0087,451.914
TBPT,,0.0088,452.270
TBPT,,0.0089,452.622
TBPT,,0.0090,452.970
TBPT,,0.0091,453.313
TBPT,,0.0092,453.653
TBPT,,0.0093,453.988
TBPT,,0.0094,454.320
TBPT,,0.0095,454.647
TBPT,,0.0096,454.971
TBPT,,0.0097,455.291
TBPT,,0.0098,455.607
TBPT,,0.0099,455.919
TBPT,,0.0100,456.228
TBPT,,0.0102,456.834
TBPT,,0.0104,457.427
TBPT,,0.0106,458.005
TBPT,,0.0108,458.569
TBPT,,0.0110,459.121
TBPT,,0.0112,459.659
TBPT,,0.0114,460.185
TBPT,,0.0116,460.698
TBPT,,0.0118,461.199
TBPT,,0.0120,461.688
TBPT,,0.0122,462.166
TBPT,,0.0124,462.633
TBPT,,0.0126,463.089
TBPT,,0.0128,463.534
TBPT,,0.0130,463.968
TBPT,,0.0132,464.392
TBPT,,0.0134,464.807
TBPT,,0.0136,465.211
TBPT,,0.0138,465.606
TBPT,,0.0140,465.992
TBPT,,0.0145,466.917
TBPT,,0.0150,467.789
TBPT,,0.0155,468.610
TBPT,,0.0160,469.384
TBPT,,0.0165,470.113
TBPT,,0.0170,470.800
TBPT,,0.0175,471.447
TBPT,,0.0180,472.057
TBPT,,0.0185,472.631
TBPT,,0.0190,473.173
TBPT,,0.0195,473.683
TBPT,,0.0200,474.163
TBPT,,0.0210,475.043
TBPT,,0.0220,475.824
TBPT,,0.0230,476.517
TBPT,,0.0240,477.132
TBPT,,0.0250,477.679
TBPT,,0.0260,478.164
TBPT,,0.0270,478.594
TBPT,,0.0280,479.976
TBPT,,0.0290,479.316
TBPT,,0.0300,479.617
TBPT,,0.0350,480.686
TBPT,,0.0400,481.275
TBPT,,0.0450,481.600

```

!20 data points

!30 data points

!40 data points

!50 data points

!60 data points

!70 data points

!80 data points

```

TBPT,,0.0500,481.780
TBPT,,0.0600,481.933
TBPT,,0.0700,481.980
TBPT,,0.0800,481.994      !90 data points
TBPT,,0.0900,481.998
TBPT,,0.1000,481.999
TBPT,,0.1070,482.000      !Last stress/strain data point

!Define plastic material properties for material 2: 100% Coarse Grain Al-5083
TB,MISO,2,1,33,0          !Defines multilinear plastic properties
TBTEMP,0                  !for 100% Coarse Grain Al-5083 using
                           !Joshi's Voce Plasticity Model
TBPT,,0.0039,282.750
TBPT,,0.0040,286.000
TBPT,,0.0045,286.683
TBPT,,0.0050,287.332
TBPT,,0.0055,287.950
TBPT,,0.0060,288.538
TBPT,,0.0065,289.067
TBPT,,0.0070,289.628
TBPT,,0.0075,290.134
TBPT,,0.0080,290.616      !10 data points
TBPT,,0.0085,291.073
TBPT,,0.0090,291.509
TBPT,,0.0095,291.923
TBPT,,0.0100,292.316
TBPT,,0.0110,293.048
TBPT,,0.0120,293.709
TBPT,,0.0130,294.308
TBPT,,0.0140,294.850
TBPT,,0.0150,295.340
TBPT,,0.0160,295.783      !20 data points
TBPT,,0.0170,296.185
TBPT,,0.0180,296.548
TBPT,,0.0190,296.876
TBPT,,0.0200,297.173
TBPT,,0.0300,298.960
TBPT,,0.0400,299.617
TBPT,,0.0500,299.859
TBPT,,0.0600,299.948
TBPT,,0.0700,299.981
TBPT,,0.0800,299.993      !30 data points
TBPT,,0.0900,299.997
TBPT,,0.1000,299.999
TBPT,,0.1080,300.000      !Last stress/strain data point

!Create dummy material 3 to change killed ufg elements to (material 1)
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,3,,72500        !Defines Young's Modulus
MPDATA,PRXY,3,,0.33       !Defines Poisson's Ratio

!Create dummy material 4 to change killed cg elements to (material 2)
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,4,,72500        !Defines Young's Modulus
MPDATA,PRXY,4,,0.33       !Defines Poisson's Ratio

!Define element to mesh with
ET,1,PLANE183             !Defines SOLID186 element
KEYOPT,1,1,0              !Indicates a full integration
KEYOPT,1,3,0              !Indicates plane strain
KEYOPT,1,6,0              !Indicates pure displacement
KEYOPT,1,10,0             !Indicates no user defined initial stress
LSEL,ALL                  !Selects all boundary lines
numb = 200                !Specifies 'numb' elements divisions per line
LESIZE,ALL,,numb          !Create 100 element division per line
ee=0.01/numb              !Computes the element size

```



```

!Mesh the model
MSHAPE,0,2D
MSHKEY,1
CM,_Y,AREA
ASEL,,,1
CM,_Y1,AREA
CHKMSH,'AREA'
CMSEL,S,_Y
AMESH,_Y1
CMDELE,_Y
CMDELE,_Y1
CMDELE,_Y2

/PREP7                                !Start the preprocessor
nn=30                                !Specifies how many coarse grains to create
!Define random locations of the centers of the coarse grains with the UFG domain
*DIM,XYZ,ARRAY,nn,3                  !Defines XYZ array with 3 elements (x,y,z)
*DIM,RL1,ARRAY,nn,3                  !Defines RL1 array with 3 elements (R,+L,-L)
*DO,ii,1,nn,1                        !Loops through and creates the required
number of volumes
    !Define the spatial locations of the 'center' of each coarse grain
    *VFILL,XYZ(ii,1),RAND,-0.0010,0.0090    !Puts a random number in 1st element of XYZ
    *VFILL,XYZ(ii,2),RAND,-0.0010,0.0090    !Puts a random number in 2nd element of XYZ
    *VFILL,XYZ(ii,3),RAND,-0.0005,0.0005    !Puts a random number in 3rd element of XYZ
    *VFILL,RL1(ii,1),RAND, 0.0002,0.0006    !Puts a random number in 1st element of RL1
    *VFILL,RL1(ii,2),RAND, 0.0010,0.0020    !Puts a random number in 2nd element of RL1
    *VFILL,RL1(ii,3),RAND, 0.0010,0.0020    !Puts a random number in 3rd element of RL1
*ENDDO

!Define random sized coarse grains at the random locations
*DO,ii,1,nn,1
    /VIEW,1,1,,                      !Changes the view to the global yz-plane
    /ANG,1
    /REP,FAST

    !Moves the origin of the working plane to XY(1),XY(2),0 and aligns the new
    !z-direction with the global x-direction
    WPLANE,1,XYZ(ii,1),XYZ(ii,2),XYZ(ii,3),XYZ(ii,1),XYZ(ii,2),XYZ(ii,3)-0.1
    !Changes the coordinate system to cylindrical coordinates
    CSWPLA,11,1

    !Pick nodes inside volume defined by XYZ and RL1
    ESEL,NONE
    NSEL,NONE
    *DO,zz,(ee/2),RL1(ii,2),ee
        rad = RL1(ii,1) - ((RL1(ii,1) - ee)/RL1(ii,2))*zz
        NSEL,A,LOC,Z,(zz - ee),(zz + ee)
        NSEL,R,LOC,X,0.0,rad
        ESLN,A,1
        EMODIF,ALL,MAT,2
    *ENDDO

    ESEL,NONE
    NSEL,NONE
    *DO,zz,-(ee/2),-RL1(ii,3),-ee
        rad = RL1(ii,1) + ((RL1(ii,1) - ee)/RL1(ii,3))*zz
        NSEL,A,LOC,Z,(zz - ee),(zz + ee)
        NSEL,R,LOC,X,0.0,rad
        ESLN,A,1
        EMODIF,ALL,MAT,2
    *ENDDO
*ENDDO

ESEL,S,MAT,,2
*GET,CG,ELEM,,COUNT
ESEL,ALL
*GET,TOT,ELEM,,COUNT
cgfrac = CG/TOT

```

```

/PNUM,KP,0
/PNUM,LINE,0
/PNUM,AREA,0
/PNUM,VOLU,0
/PNUM,NODE,0
/PNUM,TABN,0
/PNUM,SVAL,0
/NUMBER,1
/PNUM,MAT,1
EPLOT
/VIEW,1,,,1          !Changes the view to an isometric view
/ANG,1
/REP,FAST
/AUTO,1
/REP,FAST

ALLSEL,ALL
CSYS,0
FINISH
SAVE,,,ALL
FINISH

```

Local Solution

```

!Start 2D Local Fatigue Analysis
!Requires geometry, elements, and material properties be input previously
/SOLU                  !Start Solver
OUTRES,ERASE          !Erase output files
FINISH                !Exit solver
*DIM,Fname,CHAR,30,1,1,, !Create a vector named Fname
*CREATE,ansuitmp       !Create an input macro file
*VREAD,Fname,'Fname','txt',' ',IJK,30,1,1,, !Read from Fname.txt to Fname vector
(A)                   !Read input in character format
*END                  !Close the macro file
/INPUT,ansuitmp        !Switch the input file for following commands
FINISH                !Finish the file input
q = 1                 !Start the filename counter
/FILNAME,Fname(q,1,1),1 !Create output filename (b,c,d,...,txt)

!Apply boundary conditions and determine loads
/PREP7                !Start the preprocessor
!_____!
!Apply symmetry to x y and z boundaries at 0
NSEL,S,LOC,X,0.0      !Select nodes at x = 0.00
DSYM,SYMM,X           !Apply symmetry conditions
NSEL,S,LOC,Y,0.0      !Select nodes at y = 0.00
DSYM,SYMM,Y           !Apply symmetry conditions
ALLSEL,ALL            !Reselect all nodes

!Define x and y displacements from global model
tx_disp = 0.0000361   !Tensile x-disp for 3.0 mm gage length
ty_disp = -0.0000119  !Tensile y-disp for 3.0 mm gage length
cx_disp = -0.0000758  !Compressive x-disp for 3.0 mm gage length
cy_disp = 0.0000299   !Compressive y-disp for 3.0 mm gage length

FINISH
!_____!

!Start Tension Portion of First Cycle
/PREP7                !Start the preprocessor
!_____!
!Apply tensile displacements
NSEL,S,LOC,X,0.010000 !Select nodes at x = 0.01
NSEL,U,LOC,X,0.000000,0.009999 !Unselect any interior nodes
DDELE,ALL,UX,          !Delete existing x-displacements

```

```

D,ALL,UX,tx_disp                !Displace the selected nodes in tension
NSEL,S,LOC,Y,0.010000           !Select nodes at y = 0.01
NSEL,U,LOC,Y,0.000000,0.009999 !Unselect any interior nodes
DDELETE,ALL,UY,                 !Delete existing y-displacements
D,ALL,UY,ty_disp                !Displace the selected nodes in tension
ALLSEL,ALL                      !Reselect all nodes
FINISH                           !Exit the preprocessor

!Start Tensile Solution
/SOLU                            !Start the solver
NSUBST,100                       !Specify 100 Substeps
SOLVE                            !Solve the 1st tension cycle
FINISH                           !Exit the solver

!Determine and save plastic strain amplitudes every element
/POST1                           !Start the post-processor
ESEL,S,MAT,,1                   !Select all ufg elements
ETABLE,Tufg_PS,EPPL,EQV         !Create an etable of ufg plastic strains
ESEL,S,MAT,,2                   !Select all cg elements
ETABLE,Tcg_PS,EPPL,EQV         !Create an etable of cg plastic strains
ALLSEL,ALL                      !Reselect all elements

!Convert the plastic strain amplitudes to percent plastic strain amplitudes
SMULT,Tufg_PS,Tufg_PS,,100.0    !Multiply the ufg plastic strain amplitude by 100
SMULT,Tcg_PS,Tcg_PS,,100.0      !Multiply the cg plastic strain amplitude by 100
FINISH
!_____!

!Start Compression Portion of First Cycle
/PREP7                           !Start the preprocessor
!_____!
!Apply compressive displacements
NSEL,S,LOC,X,0.010000           !Select nodes at x = 0.01
NSEL,U,LOC,X,0.000000,0.009999 !Unselect any interior nodes
DDELETE,ALL,UX,                 !Delete existing x-displacements
D,ALL,UX,cx_disp                !Displace the selected nodes in compression
NSEL,S,LOC,Y,0.010000           !Select nodes at y = 0.01
NSEL,U,LOC,Y,0.000000,0.009999 !Unselect any interior nodes
DDELETE,ALL,UY,                 !Delete existing y-displacements
D,ALL,UY,cy_disp                !Displace the selected nodes in compression
ALLSEL,ALL                      !Reselect all nodes
FINISH                           !Exit the preprocessor

!Start Compressive Solution
/SOLU                            !Start the solver
NSUBST,100                       !Specify 100 substeps
SOLVE                            !Solve the 1st compression cycle
FINISH                           !Exit the solver

!Determine and save plastic strain amplitudes and stress in every element
/POST1                           !Start the post-processor
ESEL,S,MAT,,1                   !Select all ufg elements
ETABLE,Cufg_PS,EPPL,EQV         !Create an etable of ufg plastic strains
ESEL,S,MAT,,2                   !Select all cg elements
ETABLE,Ccg_PS,EPPL,EQV         !Create an etable of cg plastic strains
ALLSEL,ALL                      !Reselect all elements

!Convert the plastic strain amplitudes to percent plastic strain amplitudes
SMULT,Cufg_PS,Cufg_PS,,100.0    !Multiply the ufg plastic strain amplitude by 100
SMULT,Ccg_PS,Ccg_PS,,100.0      !Multiply the cg plastic strain amplitude by 100
FINISH
!_____!

!Determine the fatigue life and damage per cycle of every element
!Then determine the initiation life of every element, and eliminate elements accordingly
/POST1                           !Start the post-processor
!_____!
!Determine the plastic strain amplitude in every element for the first cycle

```

```

SADD,PSAm_ufg,Tufg_PS,Cufg_PS,0.5,0.5 !Average the ufg plastic strain amplitudes
SADD,PSAm_cg,Tcg_PS,Ccg_PS,0.5,0.5 !Average the cg plastic strain amplitudes

!Take the first cycle's strain amplitude to the appropriate power law exponent
SEXP,CSS_ufg,PSAm_ufg,, -1.444 !Raise ufg plastic strain amplitude to an exponent
SEXP,CSS_cg,PSAm_cg,, -1.599 !Raise cg plastic strain amplitude to an exponent

!Find the cycles to failure for every element after the first cycle
SMULT,Nf_ufg,CSS_ufg,,290.2 !Multiply the ufg result by a constant
SMULT,Nf_cg,CSS_cg,,228.08 !Multiply the cg result by a constant
SADD,Nf,Nf_ufg,Nf_cg !Add the ufg and cg cycles to failure together

!Find the damage per cycle (DPC) for every element and the size of the etable
SEXP,DPC,Nf,, -1 !Invert the cycles to failure (damage per cycle)

!Select all elements that have initial fatigue lives less than 10 cycles
init_life = 250.0 !Set the initiation life to 10 cycles
ESEL,S,ETAB,NF,1.0,init_life !Select elements that fail in less than 10 cycles

!Eliminate elements used to determine initiation life
FINISH !Exit the post-processor
/SOLU !Start the solver
EKILL,ALL !Eliminates selected elements
ALLSEL,ALL !Reselect all elements
FINISH !Exit the solver

!Change killed elements to dead ufg and cg materials
/PREP7 !Start the pre-processor
ESEL,S,LIVE !Select live elements
ESEL,INVERT !Invert the selection to eliminated elements
ESEL,R,MAT,,1 !Select killed ufg elements
EMODIF,ALL,MAT,3 !Change killed ufg elements to material 3
ESEL,S,LIVE !Select live elements
ESEL,INVERT !Invert the selection to eliminated elements
ESEL,R,MAT,,2 !Select killed cg elements
EMODIF,ALL,MAT,4 !Change killed cg elements to material 4
FINISH !Exit the pre-processor

!Calculate the residual damage
/POST1 !Start the post-processor
SMULT,RDMG,DPC,,init_life !Multiply the DPC in every element by the init. life

!Fill a vector with the residual damage
*DIM,Res_Dmg,ARRAY,num_elem !Dimension a vector to include every element
*VGET,Res_Dmg,ELEM,1,ETAB,RDMG !Fill the vector with the residual damages
FINISH !Exit the post processor
ALLSEL,ALL
EPLOT
SAVE,,,ALL !Save all database information
FINISH !Exit save
!_____!

!Loop through a number of cycles
*DO,q,2,30,1
/POST1 !Start post-processor
ETABLE,ERAS !Erase all etables
FINISH
/FILNAME,Fname(q,1,1),1 !Create output filename (b,c,d,...,txt)
FINISH

!Start Tension Portion of Cycles
/PREP7 !Start the preprocessor
!_____!
!Apply tensile displacements
NSEL,S,LOC,X,0.010000 !Select nodes at x = 0.01
NSEL,U,LOC,X,0.000000,0.009999 !Unselect any interior nodes
DDELE,ALL,UX, !Delete existing x-displacements
D,ALL,UX,tx_disp !Displace the selected nodes in tension

```

```

NSEL,S,LOC,Y,0.010000      !Select nodes at y = 0.01
NSEL,U,LOC,Y,0.000000,0.009999 !Unselect any interior nodes
DDELETE,ALL,UY,            !Delete existing y-displacements
D,ALL,UY,ty_disp           !Displace the selected nodes in tension
ALLSEL,ALL                 !Reselect all nodes
FINISH                     !Exit the preprocessor

!Start Tensile Solution
/SOLU                      !Start the solver
NSUBST,100                 !Specify 100 substeps
SOLVE                      !Solve the 1st tension cycle
FINISH                     !Exit the solver

!Determine and save plastic strain amplitudes every element
/POST1                     !Start the post-processor
ESEL,S,MAT,,1              !Select all ufg elements
ETABLE,Tufg_PS,EPPL,EQV    !Create an etable of ufg plastic strains
ESEL,S,MAT,,2              !Select all cg elements
ETABLE,Tcg_PS,EPPL,EQV     !Create an etable of cg plastic strains
ALLSEL,ALL                 !Reselect all elements

!Convert the plastic strain amplitudes to percent plastic strain amplitudes
SMULT,Tufg_PS,Tufg_PS,,100.0 !Multiply the ufg plastic strain amplitude by 100
SMULT,Tcg_PS,Tcg_PS,,100.0   !Multiply the cg plastic strain amplitude by 100
!_____!

!Start Compression Portion of Cycles
/PREP7                     !Start the preprocessor
!_____!
!Apply compressive displacements
NSEL,S,LOC,X,0.010000      !Select nodes at x = 0.01
NSEL,U,LOC,X,0.000000,0.009999 !Unselect any interior nodes
DDELETE,ALL,UX,            !Delete existing x-displacements
D,ALL,UX,cx_disp           !Displace the selected nodes in compression
NSEL,S,LOC,Y,0.010000      !Select nodes at y = 0.01
NSEL,U,LOC,Y,0.000000,0.009999 !Unselect any interior nodes
DDELETE,ALL,UY,            !Delete existing y-displacements
D,ALL,UY,cy_disp           !Displace the selected nodes in compression
ALLSEL,ALL                 !Reselect all nodes
FINISH

!Start Compressive Solution
/SOLU                      !Start the solver
NSUBST,100                 !Specify 100 substeps
SOLVE                      !Solve the 1st compression cycle
FINISH                     !Exit the solver

!Determine and save plastic strain amplitudes and stress in every element
/POST1                     !Start the post-processor
ESEL,S,MAT,,1              !Select all ufg elements
ETABLE,Cufg_PS,EPPL,EQV    !Create an etable of ufg plastic strains
ESEL,S,MAT,,2              !Select all cg elements
ETABLE,Ccg_PS,EPPL,EQV     !Create an etable of cg plastic strains
ALLSEL,ALL                 !Reselect all elements

!Convert the plastic strain amplitudes to percent plastic strain amplitudes
SMULT,Cufg_PS,Cufg_PS,,100.0 !Multiply the ufg plastic strain amplitude by 100
SMULT,Ccg_PS,Ccg_PS,,100.0   !Multiply the cg plastic strain amplitude by 100
!_____!

!Find the fatigue life and damage per cycle of every element
!Then determine the accumulated damage of every element and eliminate those
!That exceed unity
/POST1                     !Start the post-processor
!_____!
!Determine the plastic strain amplitude in every element for the first cycle
SADD,PSAm_ufg,Tufg_PS,Cufg_PS,0.5,0.5 !Average the ufg plastic strain amplitudes
SADD,PSAm_cg,Tcg_PS,Ccg_PS,0.5,0.5     !Average the cg plastic strain amplitudes

```

```

!Take the first cycle's strain amplitude to the appropriate power law exponent
SEXP,CSS_ufg,PSAm_ufg,, -1.444      !Raise the ufg plastic strain amplitude to a
exponent
SEXP,CSS_cg,PSAm_cg,, -1.599        !Raise the cg plastic strain amplitude to an
exponent

!Find the cycles to failure for every element after the first cycle
SMULT,Nf_ufg,CSS_ufg,,290.2          !Multiply the ufg result by a constant
SMULT,Nf_cg,CSS_cg,,228.08           !Multiply the cg result by a constant
SADD,Nf,Nf_ufg,Nf_cg                 !Add the ufg and cg cycles to failure together

!Find the damage per cycle (DPC) for every element and define number of simulated cycles
SEXP,DPC,Nf,, -1                     !Invert the cycles to failure (damage per cycle)
num_cycles = 200                     !Specify the number of cycles represented by this load step

!Calculate the damage incurred for this load step
SMULT,CDMG,DPC,,num_cycles            !Multiply the DPC by the number of cycles
SMULT,PDMG,DPC,,num_cycles            !Create dummy etable with same values

!Add the damage accrued here to the residual damage
*VPUT,Res_Dmg,ELEM,,ETAB,PDMG        !Put the Residual damage array back into an etable
SADD,RDMG,CDMG,PDMG                  !Add the current damage to the residual damage
*VGET,Res_Dmg,ELEM,1,ETAB,RDMG       !Put the Residual damage back into the array

!Choose elements that have exceeded the damage criteria and eliminate
ESEL,S,ETAB,RDMG,1.0,                !Select elements with damage greater than unity
FINISH                                !Exit the Post Processor
/SOLU                                 !Start the solver
EKILL,ALL                             !Eliminate selected elements
ALLSEL,ALL                            !Reselect all elements
FINISH                                !Exit Solver

!Change killed elements to dead ufg and cg materials
/PREP7                                !Start the pre-processor
ESEL,S,LIVE                           !Select live elements
ESEL,INVERT                           !Invert the selection to eliminated elements
ESEL,R,MAT,,1                         !Select killed ufg elements
EMODIF,ALL,MAT,3                      !Change killed ufg elements to material 3
ESEL,S,LIVE                           !Select live elements
ESEL,INVERT                           !Invert the selection to eliminated elements
ESEL,R,MAT,,2                         !Select killed cg elements
EMODIF,ALL,MAT,4                      !Change killed cg elements to material 4
FINISH                                !Exit the pre-processor

ALLSEL,ALL
EPLOT
SAVE,,,ALL                            !Save the current solution to b,c,d,...,txt
FINISH
!
*ENDDO

```

Cyclic 3D Input Files

Global Model

```

!Builds the 3D global fatigue model
!All dimensions are in millimeters
/FILNAME,model,0
/CWD,'C:\ANSYS\3D Fatigue 3D Global'
/PREP7                                !Starts ANSYS

!Define geometry
K,1,0.00,0.00,0.00                   !Define keypoint 01 at (0.00,0.00,0.00)
K,2,3.75,0.00,0.00                   !Define keypoint 02 at (3.75,0.00,0.00)

```

```

K,3,5.00,1.25,0.00      !Define keypoint 03 at (5.00,1.25,0.00)
K,4,6.25,0.00,0.00      !Define keypoint 04 at (6.25,0.00,0.00)
K,5,8.00,0.00,0.00      !Define keypoint 05 at (8.00,0.00,0.00)
K,6,8.00,3.00,0.00      !Define keypoint 06 at (8.00,3.00,0.00)
K,7,3.00,3.00,0.00      !Define keypoint 07 at (3.00,3.00,0.00)
K,8,3.00,0.50,0.00      !Define keypoint 08 at (3.00,0.50,0.00)
K,9,0.00,0.50,0.00      !Define keypoint 09 at (0.00,0.50,0.00)

L,1,2                    !Define line 01 between points 1 and 2
LARC,2,4,3,1.25          !Define line 02 as arc with rad 1.25 between points 2,3,4
L,4,5                    !Define line 03 between points 4 and 5
L,5,6                    !Define line 04 between points 5 and 6
L,6,7                    !Define line 05 between points 6 and 7
LFILLT,4,5,1.25          !Define line 06 (fillet) with rad 1.25 between lines 4 & 5
LCOMB,4,6                 !Define line 04 by combining lines 4 and 6 (06 is deleted)
L,7,8                    !Define line 06 between points 7 and 8
L,8,9                    !Define line 07 between points 8 and 9
LFILLT,6,7,1.25          !Define line 08 (fillet) with rad 1.25 between lines 7 & 8
LCOMB,6,8                 !Define line 06 by combining lines 6 and 8 (08 is deleted)
L,9,1                    !Define line 08 between points 9 and 1
AL,1,2,3,4,5,6,7,8       !Creates area from lines 1, 2, 3, 4, 5, 6, 7, and 8
VOFFST,1,0.5             !Creates volume from area 1 by offsetting 0.5 mm in +z-dir

!Define global model as 15% cg content from Joshi's model
MPTEMP,,,,,,,,           !Defines elastic material properties
MPTEMP,1,0
MPDATA,EX,1,,72500       !Defines Young's Modulus
MPDATA,PRXY,1,,0.33      !Defines Poisson's Ratio

TB,MISO,1,1,73,0         !Defines multilinear plastic properties
TBTEMP,0
TBPT,,0.0052,377.000
TBPT,,0.0053,380.100
TBPT,,0.0054,381.077
TBPT,,0.0055,382.040
TBPT,,0.0056,382.990
TBPT,,0.0057,383.927
TBPT,,0.0058,384.850
TBPT,,0.0059,385.761
TBPT,,0.0060,386.659
TBPT,,0.0061,387.544     !10 data points
TBPT,,0.0062,388.417
TBPT,,0.0063,389.278
TBPT,,0.0064,390.126
TBPT,,0.0065,390.963
TBPT,,0.0066,391.789
TBPT,,0.0067,392.602
TBPT,,0.0068,393.405
TBPT,,0.0069,394.196
TBPT,,0.0070,394.976
TBPT,,0.0071,395.745     !20 data points
TBPT,,0.0072,396.503
TBPT,,0.0073,397.251
TBPT,,0.0074,397.989
TBPT,,0.0075,398.716
TBPT,,0.0076,399.433
TBPT,,0.0077,400.140
TBPT,,0.0078,400.837
TBPT,,0.0079,401.524
TBPT,,0.0080,402.202
TBPT,,0.0082,403.529     !30 data points
TBPT,,0.0084,404.219
TBPT,,0.0086,406.074
TBPT,,0.0088,407.294
TBPT,,0.0090,408.480
TBPT,,0.0092,409.633
TBPT,,0.0094,410.754
TBPT,,0.0096,411.844

```

```

TBPT,,0.0098,412.904
TBPT,,0.0100,413.934
TBPT,,0.0105,416.386      !40 data points
TBPT,,0.0110,418.672
TBPT,,0.0115,420.803
TBPT,,0.0120,422.789
TBPT,,0.0125,424.641
TBPT,,0.0130,426.366
TBPT,,0.0135,427.974
TBPT,,0.0140,429.473
TBPT,,0.0145,430.871
TBPT,,0.0150,432.173
TBPT,,0.0155,433.387      !50 data points
TBPT,,0.0160,434.519
TBPT,,0.0165,435.573
TBPT,,0.0170,436.556
TBPT,,0.0175,437.473
TBPT,,0.0180,438.327
TBPT,,0.0190,439.865
TBPT,,0.0200,441.201
TBPT,,0.0210,442.362
TBPT,,0.0220,443.371
TBPT,,0.0230,444.247      !60 data points
TBPT,,0.0240,445.009
TBPT,,0.0250,445.670
TBPT,,0.0300,447.882
TBPT,,0.0350,448.977
TBPT,,0.0400,449.519
TBPT,,0.0450,449.787
TBPT,,0.0500,449.918
TBPT,,0.0600,450.018
TBPT,,0.0700,450.042
TBPT,,0.0800,450.048      !70 data points
TBPT,,0.0900,450.050
TBPT,,0.1000,450.050
TBPT,,0.1100,450.050      !Last stress/strain data point

ET,2,SOLID186              !Defines element to mesh with
LSEL,S,LOC,X,0.0
LSEL,A,LINE,,18,23
LESIZE,ALL,,5
LSEL,ALL
LESIZE,01,0.1
LESIZE,09,0.1
LESIZE,07,0.1
LESIZE,15,0.1
LESIZE,02,,18
LESIZE,10,,18
LESIZE,06,,18
LESIZE,14,,18
LESIZE,03,,7
LESIZE,11,,7
LESIZE,04,,12
LESIZE,12,,12
LESIZE,05,,15
LESIZE,13,,15

CM,_Y,VOLU                 !Makes a volume group named _Y
VSEL,,,1                   !Selects volume 1
CM,_Y1,VOLU                !Makes a volume group named _Y1
CHKMSH,'VOLU'              !Checks the volume for previous meshes
CMSEL,S,_Y                  !Selects volume 1 to be meshed
VSWEEP,_Y1                  !Meshes volume 1 using swept mapping
CMDELE,_Y                   !Deletes volume group _Y
CMDELE,_Y1                  !Deletes volume group _Y1
CMDELE,_Y2                  !Deletes volume group _Y2

ALLSEL,ALL                  !Reselect all nodes

```



```

FINISH                                !Exit the preprocessor
SAVE,,,ALL                           !Save the model and the constraints

```

Global Solution

```

!Start 3D global fatigue analysis
!Requires geometry, elements and material properties be input previously
!Apply boundary conditions and determine loads
/FILNAME,constraints,0               !Specify filename
/PREP7                               !Start preprocessor
!_____!
!Apply symmetry to x, y and z boundaries at 0
NSEL,S,LOC,X,0.0                     !Select all nodes on yz-plane at x = 0.0
DSYM,SYMM,X                           !Apply symmetry conditions to selected nodes
NSEL,S,LOC,Y,0.0                     !Select all nodes on xz-plane at y = 0.0
DSYM,SYMM,Y                           !Apply symmetry conditions to selected nodes
NSEL,S,LOC,Z,0.0                     !Select all nodes on xy-plane at z = 0.0
DSYM,SYMM,Z                           !Apply symmetry conditions to selected nodes

!Determine appropriate displacement from plastic strain amplitude
strn_amp = 0.001                     !Sets the constant plastic strain amplitude
gage_len = 3.00                      !sets the gage length
x_disp = gage_len*(590.0/72500.0 + strn_amp) !Determines the required displacement
ALLSEL,ALL                           !Reselect all nodes
FINISH                               !Exit preprocessor
SAVE,,,ALL                           !Save all database information
!_____!

```

```

!Start Tension Portion of First Cycle
/FILNAME,tension_cycle_01,1          !Specify Filename
/PREP7                               !Start preprocessor
!_____!
!Apply tension displacement
ASEL,S,AREA,,4                       !Select the pinhole surface
NSLA,,1                             !Select nodes on pinhole surface
DDELE,ALL,UX                         !Delete existing displacements
NSEL,R,LOC,X,5.0,6.25                !Select nodes at top of pinhole
D,ALL,UX,x_disp                      !Displace the selected nodes
ALLSEL,ALL                           !Reselect all nodes
FINISH                               !Exit the preprocessor

!Start tensile solution
/SOLU                                !Start the solver
SOLVE                                !Solve the load step
FINISH                               !Exit the solver
SAVE,,,ALL                           !Save all database information
FINISH
!_____!

```

```

!Start Compression Portion of First Cycle
/FILNAME,compress_cycle01,1          !Specify Filename
/PREP7                               !Start Preprocessor
!_____!
!Apply Compression displacement
ASEL,S,AREA,,4                       !Select the pinhole surface
NSLA,,1                             !Select nodes on pinhole surface
DDELE,ALL,UX                         !Delete existing displacements
NSEL,R,LOC,X,3.75,5.00               !Select nodes at bottom of pinhole
D,ALL,UX,-x_disp                     !Displace the selected nodes
ALLSEL,ALL                           !Reselect all nodes
FINISH                               !Exit the preprocessor

!Start Compressive solution
/SOLU                                !Start the solver

```

```

SOLVE                !Solve the load step
FINISH               !Exit the solver
SAVE,,,,ALL          !Save all database information
FINISH

```

Local Model

```

!Fatigue model containing geometry, material, and element information for
!Fatigue model - All dimensions are in millimeters
/FILNAME,model,0
/CWD,'C:\ANSYS\3D Fatigue Local 20x20x20'

/PREP7                !Starts preprocessor
!Define geometry
K,1,0.01,0.01,0.01    !Define keypoint 1 at (0.01,0.01,0.01) mm
K,2,0.01,0.00,0.01    !Define keypoint 2 at (0.01,0.00,0.01) mm
K,3,0.00,0.00,0.01    !Define keypoint 3 at (0.00,0.00,0.01) mm
K,4,0.00,0.01,0.01    !Define keypoint 4 at (0.00,0.01,0.01) mm
A,1,2,3,4              !Define area 1 from points 1-4
VOFFST,1,0.01          !Define volume 1 by extruding area 1 10.0 microns

!Defines elastic material properties for material 1: 0.0% Coarse Grain Al-5083
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,72500      !Defines Young's Modulus
MPDATA,PRXY,1,,0.33     !Defines Poisson's Ratio

!Defines elastic material properties for material 2: 100% Coarse Grain Al-5083
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,2,,72500      !Defines Young's Modulus
MPDATA,PRXY,2,,0.33     !Defines Poisson's Ratio

!Defines plastic material properties for material 1: 0.0% Coarse Grain Al-5083
TB,MISO,1,1,93,0        !Defines multilinear plastic properties
TBTEMP,0                !for 0% Coarse Grain Al-5083 using
TBPT,,0.0060,435.000    !Joshi's Voce Plasticity Model
TBPT,,0.0061,441.000
TBPT,,0.0062,441.485
TBPT,,0.0063,441.965
TBPT,,0.0064,442.438
TBPT,,0.0065,442.907
TBPT,,0.0066,443.369
TBPT,,0.0067,443.826
TBPT,,0.0068,444.278
TBPT,,0.0069,444.725    !10 data points
TBPT,,0.0070,445.166
TBPT,,0.0071,445.602
TBPT,,0.0072,446.032
TBPT,,0.0073,446.458
TBPT,,0.0074,446.879
TBPT,,0.0075,447.294
TBPT,,0.0076,447.705
TBPT,,0.0077,448.111
TBPT,,0.0078,448.512
TBPT,,0.0079,448.908    !20 data points
TBPT,,0.0080,449.230
TBPT,,0.0081,449.687
TBPT,,0.0082,450.069
TBPT,,0.0083,450.447
TBPT,,0.0084,450.820
TBPT,,0.0085,451.189
TBPT,,0.0086,451.554
TBPT,,0.0087,451.914
TBPT,,0.0088,452.270
TBPT,,0.0089,452.622    !30 data points

```

```

TBPT,,0.0090,452.970
TBPT,,0.0091,453.313
TBPT,,0.0092,453.653
TBPT,,0.0093,453.988
TBPT,,0.0094,454.320
TBPT,,0.0095,454.647
TBPT,,0.0096,454.971
TBPT,,0.0097,455.291
TBPT,,0.0098,455.607
TBPT,,0.0099,455.919
TBPT,,0.0100,456.228
TBPT,,0.0102,456.834
TBPT,,0.0104,457.427
TBPT,,0.0106,458.005
TBPT,,0.0108,458.569
TBPT,,0.0110,459.121
TBPT,,0.0112,459.659
TBPT,,0.0114,460.185
TBPT,,0.0116,460.698
TBPT,,0.0118,461.199
TBPT,,0.0120,461.688
TBPT,,0.0122,462.166
TBPT,,0.0124,462.633
TBPT,,0.0126,463.089
TBPT,,0.0128,463.534
TBPT,,0.0130,463.968
TBPT,,0.0132,464.392
TBPT,,0.0134,464.807
TBPT,,0.0136,465.211
TBPT,,0.0138,465.606
TBPT,,0.0140,465.992
TBPT,,0.0145,466.917
TBPT,,0.0150,467.789
TBPT,,0.0155,468.610
TBPT,,0.0160,469.384
TBPT,,0.0165,470.113
TBPT,,0.0170,470.800
TBPT,,0.0175,471.447
TBPT,,0.0180,472.057
TBPT,,0.0185,472.631
TBPT,,0.0190,473.173
TBPT,,0.0195,473.683
TBPT,,0.0200,474.163
TBPT,,0.0210,475.043
TBPT,,0.0220,475.824
TBPT,,0.0230,476.517
TBPT,,0.0240,477.132
TBPT,,0.0250,477.679
TBPT,,0.0260,478.164
TBPT,,0.0270,478.594
TBPT,,0.0280,479.976
TBPT,,0.0290,479.316
TBPT,,0.0300,479.617
TBPT,,0.0350,480.686
TBPT,,0.0400,481.275
TBPT,,0.0450,481.600
TBPT,,0.0500,481.780
TBPT,,0.0600,481.933
TBPT,,0.0700,481.980
TBPT,,0.0800,481.994
TBPT,,0.0900,481.998
TBPT,,0.1000,481.999
TBPT,,0.1070,482.000

!40 data points

!50 data points

!60 data points

!70 data points

!80 data points

!90 data points

!Last stress/strain data point

!Define plastic material properties for material 2: 100% Coarse Grain Al-5083
TB,MISO,2,1,33,0
TBTEMP,0
TBPT,,0.0039,282.750
!Defines multilinear plastic properties
!for 100% Coarse Grain Al-5083 using
!Joshi's Voce Plasticity Model

```

```

TBPT,,0.0040,286.000
TBPT,,0.0045,286.683
TBPT,,0.0050,287.332
TBPT,,0.0055,287.950
TBPT,,0.0060,288.538
TBPT,,0.0065,289.067
TBPT,,0.0070,289.628
TBPT,,0.0075,290.134
TBPT,,0.0080,290.616      !10 data points
TBPT,,0.0085,291.073
TBPT,,0.0090,291.509
TBPT,,0.0095,291.923
TBPT,,0.0100,292.316
TBPT,,0.0110,293.048
TBPT,,0.0120,293.709
TBPT,,0.0130,294.308
TBPT,,0.0140,294.850
TBPT,,0.0150,295.340
TBPT,,0.0160,295.783      !20 data points
TBPT,,0.0170,296.185
TBPT,,0.0180,296.548
TBPT,,0.0190,296.876
TBPT,,0.0200,297.173
TBPT,,0.0300,298.960
TBPT,,0.0400,299.617
TBPT,,0.0500,299.859
TBPT,,0.0600,299.948
TBPT,,0.0700,299.981
TBPT,,0.0800,299.993      !30 data points
TBPT,,0.0900,299.997
TBPT,,0.1000,299.999
TBPT,,0.1080,300.000      !Last stress/strain data point

!Create dummy material 3 to change killed ufg elements to (material 1)
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,3,,72500      !Defines Young's Modulus
MPDATA,PRXY,3,,0.33      !Defines Poisson's Ratio

!Create dummy material 4 to change killed cg elements to (material 2)
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,4,,72500      !Defines Young's Modulus
MPDATA,PRXY,4,,0.33      !Defines Poisson's Ratio

!Define element to mesh with
ET,2,SOLID186      !Defines SOLID186 element
LSEL,S,,1,12      !Selects all ufg boundary lines (lines 1-12)
numb = 20      !Specifies 'numb' elements divisions per line
LESIZE,ALL,,numb      !Applies 'numb' element divisions per line
ee=0.01/numb      !Computes the element size

CM,_Y,VOLU      !Makes a volume group named _Y
VSEL,,,1      !Selects all volumes
CM,_Y1,VOLU      !Makes a volume group named _Y1
CHKMSH,'VOLU'      !Checks the volume for previous meshes
CMSEL,S,_Y      !Selects volume 1 to be meshed
VSWEEP,_Y1      !Meshes volume 1 using swept mapping
CMDELE,_Y      !Deletes volume group _Y
CMDELE,_Y1      !Deletes volume group _Y1
CMDELE,_Y2      !Deletes volume group _Y2

/PREP7      !Start the preprocessor
*DIM,XYZ,ARRAY,2,3      !Defines a XYZ array with 3 elements
(xloc,yloc,zloc)
*DIM,RL,ARRAY,2,2
*VFILL,XYZ(1,1),DATA,0.0
*VFILL,XYZ(1,2),DATA,0.0

```

```

*VFILL,XYZ(1,3),DATA,0.0
*VFILL,XYZ(2,1),DATA,0.005
*VFILL,XYZ(2,2),DATA,0.006
*VFILL,XYZ(2,3),DATA,0.006
Rin = ee
*VFILL,RL(1,1),DATA,0.003
*VFILL,RL(1,2),DATA,0.014
*VFILL,RL(2,1),DATA,0.0027
*VFILL,RL(2,2),DATA,0.006

*DO,ii,1,2,1
    /VIEW,1,1,, !Changes the view to the global yz-plane
    /ANG,1
    /REP,FAST

    !Moves the origin of the working plane to XYZ(1),XYZ(2),XYZ(3) and aligns the new
    !z-direction with the global x-direction
    WPLANE,1,XYZ(ii,1),XYZ(ii,2),XYZ(ii,3),XYZ(ii,1),XYZ(ii,2),XYZ(ii,3)-0.1
    !Changes the coordinate system to cylindrical coordinates
    CSWPLA,11,1

    ESEL,NONE
    NSEL,NONE
    *DO,zz,(ee/2),RL(ii,2),ee
        rad = RL(ii,1) - ((RL(ii,1) - Rin)/RL(ii,2))*zz
        NSEL,A,LOC,Z,(zz - ee),(zz + ee)
        NSEL,R,LOC,X,0.0,rad
        ESLN,A,1
        EMODIF,ALL,MAT,2
    *ENDDO

    ESEL,NONE
    NSEL,NONE
    *DO,zz,-(ee/2),-RL(ii,2),-ee
        rad = RL(ii,1) + ((RL(ii,1) - Rin)/RL(ii,2))*zz
        NSEL,A,LOC,Z,(zz - ee),(zz + ee)
        NSEL,R,LOC,X,0.0,rad
        ESLN,A,1
        EMODIF,ALL,MAT,2
    *ENDDO
*ENDDO

ESEL,S,MAT,,2
*GET,CG,ELEM,,COUNT
ESEL,ALL
*GET,TOT,ELEM,,COUNT
cgfrac = CG/TOT

EPLLOT
/VIEW,1,1,1,1 !Changes the view to an isometric view
/ANG,1
/REP,FAST
ALLSEL,ALL
CSYS,0
FINISH
SAVE,,,ALL
FINISH

/POST1
/PNUM,KP,0
/PNUM,LINE,0
/PNUM,AREA,0
/PNUM,VOLU,0
/PNUM,NODE,0
/PNUM,TABN,0
/PNUM,SVAL,0
/NUMBER,1
/PNUM,MAT,1

```

```
/REPLOT
esel,s,mat,,2
epplot
```

Local Solution

```
!Start 3D Local Fatigue Analysis
!Requires geometry, elements, and material properties be input previously
/SOLU                                !Start Solver
OUTRES,ERASE                         !Erase output files
FINISH                              !Exit solver
*DIM,Fname,CHAR,30,1,1,,            !Create a vector named Fname
*CREATE,ansuitmp                     !Create an input macro file
*VREAD,Fname,'Fname','txt',' ',IJK,30,1,1,, !Read from Fname.txt to Fname vector
(A)                                 !Read input in character format
*END                                 !Close the macro file
/INPUT,ansuitmp                     !Switch the input file for following commands
FINISH                              !Finish the file input
q = 1                               !Start the filename counter
/FILNAME,Fname(q,1,1),1             !Create output filename (b,c,d,...,txt)

!Apply boundary conditions and determine loads
/PREP7                              !Start the preprocessor
!_____!
!Apply symmetry to x y and z boundaries at 0
NSEL,S,LOC,X,0.0                    !Select nodes at x = 0.00
DSYM,SYMM,X                         !Apply symmetry conditions
NSEL,S,LOC,Y,0.0                    !Select nodes at y = 0.00
DSYM,SYMM,Y                         !Apply symmetry conditions
NSEL,S,LOC,Z,0.0                    !Select nodes at z = 0.00
DSYM,SYMM,Z                         !Apply symmetry conditions
ALLSEL,ALL                          !Reselect all nodes

!Define x and y displacements from global models
tx_disp = 0.0000457
ty_disp = -0.0000151
tz_disp = -0.0000151
cx_disp = -0.0001080
cy_disp = 0.0000444
cz_disp = 0.0000443
FINISH
!_____!

!Start Tension Portion of First Cycle
/PREP7                              !Start the preprocessor
!_____!
!Apply tensile displacement
NSEL,S,LOC,X,0.01                   !Select nodes at x = 0.01
DDELE,ALL,UX,                       !Delete existing displacements
D,ALL,UX,tx_disp                     !Displace the selected nodes in tension
NSEL,S,LOC,Y,0.01                   !Select nodes at y = 0.01
DDELE,ALL,UY,                       !Delete existing displacements
D,ALL,UY,ty_disp                     !Displace the selected nodes
NSEL,S,LOC,Z,0.01                   !Select nodes at z = 0.01
DDELE,ALL,UZ,                       !Delete existing displacements
D,ALL,UZ,tz_disp                     !Displace the selected nodes
ALLSEL,ALL                          !Select all nodes
FINISH                              !Exit the preprocessor

!Start Tensile Solution
/SOLU                                !Start the solver
NSUBST,100                          !Specify 100 Substeps
SOLVE                                !Solve the 1st tension cycle
FINISH                              !Exit the solver

!Determine and save plastic strain amplitudes every element
```

```

/POST1                                !Start the post-processor
ESEL,S,MAT,,1                         !Select all ufg elements
ETABLE,Tufg_PS,EPPL,EQV               !Create an etable of ufg plastic strains
ESEL,S,MAT,,2                         !Select all cg elements
ETABLE,Tcg_PS,EPPL,EQV               !Create an etable of cg plastic strains
ALLSEL,ALL                           !Reselect all elements

!Convert the plastic strain amplitudes to percent plastic strain amplitudes
SMULT,Tufg_PS,Tufg_PS,,100.0         !Multiply the ufg plastic strain amplitude by 100
SMULT,Tcg_PS,Tcg_PS,,100.0           !Multiply the cg plastic strain amplitude by 100

FINISH
!_____!

!Start Compression Portion of First Cycle
/PREP7                                !Start the preprocessor
!_____!
NSEL,S,LOC,X,0.01                     !Select nodes at x = 0.01
DDELE,ALL,UX,                         !Delete existing displacements
D,ALL,UX,cx_disp                      !Displace the selected nodes in tension
NSEL,S,LOC,Y,0.01                     !Select nodes at y = 0.01
DDELE,ALL,UY,                         !Delete existing displacements
D,ALL,UY,cy_disp                      !Displace the selected nodes
NSEL,S,LOC,Z,0.01                     !Select nodes at z = 0.01
DDELE,ALL,UZ,                         !Delete existing displacements
D,ALL,UZ,cz_disp                      !Displace the selected nodes
ALLSEL,ALL                           !Reselect all nodes
FINISH                                !Exit the preprocessor

!Start Compressive Solution
/SOLU                                  !Start the solver
NSUBST,100                            !Specify 100 Substeps
SOLVE                                  !Solve the 1st compression cycle
FINISH                                !Exit the solver

!Determine and save plastic strain amplitudes and stress in every element
/POST1                                !Start the post-processor
ESEL,S,MAT,,1                         !Select all ufg elements
ETABLE,Cufg_PS,EPPL,EQV               !Create an etable of ufg plastic strains
ESEL,S,MAT,,2                         !Select all cg elements
ETABLE,Ccg_PS,EPPL,EQV               !Create an etable of cg plastic strains
ALLSEL,ALL                           !Reselect all elements

!Convert the plastic strain amplitudes to percent plastic strain amplitudes
SMULT,Cufg_PS,Cufg_PS,,100.0         !Multiply the ufg plastic strain amplitude by 100
SMULT,Ccg_PS,Ccg_PS,,100.0           !Multiply the cg plastic strain amplitude by 100
FINISH
!_____!

!Determine the fatigue life and damage per cycle of every element
!Then determine the initiation life of every element, and eliminate elements accordingly
/POST1                                !Start the post-processor
!_____!
!Determine the plastic strain amplitude in every element for the first cycle
SADD,PSAm_ufg,Tufg_PS,Cufg_PS,0.5,0.5 !Average the ufg plastic strain amplitudes
SADD,PSAm_cg,Tcg_PS,Ccg_PS,0.5,0.5    !Average the cg plastic strain amplitudes

!Take the first cycle's strain amplitude to the appropriate power law exponent
SEXP,CSS_ufg,PSAm_ufg,,-1.444         !Raise ufg plastic strain amplitude to an exponent
SEXP,CSS_cg,PSAm_cg,,-1.599          !Raise cg plastic strain amplitude to an exponent

!Find the cycles to failure for every element after the first cycle
SMULT,Nf_ufg,CSS_ufg,,290.2           !Multiply the ufg result by a constant
SMULT,Nf_cg,CSS_cg,,228.08            !Multiply the cg result by a constant
SADD,Nf,Nf_ufg,Nf_cg                 !Add the ufg and cg cycles to failure together

!Find the damage per cycle (DPC) for every element and the size of the etable
SEXP,DPC,Nf,,-1                       !Invert the cycles to failure (damage per cycle)

```



```

NSUBST,100                !Specify 100 substeps
SOLVE                    !Solve the 1st tension cycle
FINISH                   !Exit the solver

!Determine and save plastic strain amplitudes every element
/POST1                  !Start the post-processor
ESEL,S,MAT,,1           !Select all ufg elements
ETABLE,Tufg_PS,EPPL,EQV !Create an etable of ufg plastic strains
ESEL,S,MAT,,2           !Select all cg elements
ETABLE,Tcg_PS,EPPL,EQV  !Create an etable of cg plastic strains
ALLSEL,ALL              !Reselect all elements

!Convert the plastic strain amplitudes to percent plastic strain amplitudes
SMULT,Tufg_PS,Tufg_PS,,100.0 !Multiply the ufg plastic strain amplitude by 100
SMULT,Tcg_PS,Tcg_PS,,100.0   !Multiply the cg plastic strain amplitude by 100
!_____!

!Start Compression Portion of Each Cycle
/PREP7                  !Start the preprocessor
!_____!
NSEL,S,LOC,X,0.01       !Select nodes at x = 0.01
DDELE,ALL,UX            !Delete existing displacements
D,ALL,UX,cx_disp        !Displace the selected nodes in compression
NSEL,S,LOC,Y,0.01       !Select nodes at y = 0.01
DDELE,ALL,UY            !Delete existing displacements
D,ALL,UY,cy_disp        !Displace the selected nodes in compression
NSEL,S,LOC,Z,0.01       !Select nodes at z = 0.01
DDELE,ALL,UZ            !Delete existing displacements
D,ALL,UZ,cz_disp        !Displace the selected nodes in compression
ALLSEL,ALL              !Reselect all nodes
FINISH                  !Exit the preprocessor

!Start Compressive Solution
/SOLU                   !Start the solver
NSUBST,100              !Specify 100 substeps
SOLVE                   !Solve the 1st compression cycle
FINISH                  !Exit the solver

!Determine and save plastic strain amplitudes and stress in every element
/POST1                  !Start the post-processor
ESEL,S,MAT,,1           !Select all ufg elements
ETABLE,Cufg_PS,EPPL,EQV !Create an etable of ufg plastic strains
ESEL,S,MAT,,2           !Select all cg elements
ETABLE,Ccg_PS,EPPL,EQV  !Create an etable of cg plastic strains
ALLSEL,ALL              !Reselect all elements

!Convert the plastic strain amplitudes to percent plastic strain amplitudes
SMULT,Cufg_PS,Cufg_PS,,100.0 !Multiply the ufg plastic strain amplitude by 100
SMULT,Ccg_PS,Ccg_PS,,100.0   !Multiply the cg plastic strain amplitude by 100
!_____!

!Find the fatigue life and damage per cycle of every element
!Then determine the accumulated damage of every element and eliminate those
!That exceed unity
/POST1                  !Start the post-processor
!_____!
!Determine the plastic strain amplitude in every element for the first cycle
SADD,PSAm_ufg,Tufg_PS,Cufg_PS,0.5,0.5 !Average the ufg plastic strain amplitudes
SADD,PSAm_cg,Tcg_PS,Ccg_PS,0.5,0.5     !Average the cg plastic strain amplitudes

!Take the first cycle's strain amplitude to the appropriate power law exponent
SEXP,CSS_ufg,PSAm_ufg,-1.444 !Raise ufg plastic strain amplitude to a exponent
SEXP,CSS_cg,PSAm_cg,-1.599   !Raise cg plastic strain amplitude to an exponent

!Find the cycles to failure for every element after the first cycle
SMULT,Nf_ufg,CSS_ufg,,290.2 !Multiply the ufg result by a constant
SMULT,Nf_cg,CSS_cg,,228.08  !Multiply the cg result by a constant
SADD,Nf,Nf_ufg,Nf_cg        !Add the ufg and cg cycles to failure together

```

```

!Find the damage per cycle (DPC) for every element and define a number of representative
cycles
SEXP,DPC,Nf,, -1                !Invert the cycles to failure (damage per cycle)
num_cycles = 200                !Specify number of cycles to be 250

!Calculate the damage incurred for this load step
SMULT,CDMG,DPC,,num_cycles      !Multiply the DPC by the number of cycles
SMULT,PDMG,DPC,,num_cycles      !Create an equivalent dummy etable

!Add the damage accrued here to the residual damage
*VPUT,Res_Dmg,ELEM,,ETAB,PDMG   !Put the Residual damage array back into an etable
SADD,RDMG,CDMG,PDMG             !Add the current damage to the residual damage
*VGET,Res_Dmg,ELEM,1,ETAB,RDMG  !Put the Residual damage back into the array

!Choose elements that have exceeded the damage criteria and eliminate
ESEL,S,ETAB,RDMG,1.0            !Select elements with damage greater than unity
FINISH                          !Exit the Post Processor
/SOLU                           !Start the solver
EKILL,ALL                      !Eliminate selected elements
ALLSEL,ALL                     !Reselect all elements
FINISH                          !Exit Solver

!Change killed elements to dead ufg and cg materials
/PREP7                          !Start the pre-processor
ESEL,S,LIVE                    !Select live elements
ESEL,INVERT                    !Invert the selection to eliminated elements
ESEL,R,MAT,,1                 !Select killed ufg elements
EMODIF,ALL,MAT,3              !Change killed ufg elements to material 3
ESEL,S,LIVE                    !Select live elements
ESEL,INVERT                    !Invert the selection to eliminated elements
ESEL,R,MAT,,2                 !Select killed cg elements
EMODIF,ALL,MAT,4              !Change killed cg elements to material 4
FINISH                          !Exit the pre-processor
ALLSEL,ALL
EPLOT
SAVE,,,,,ALL                  !Save the current solution to b,c,d,,,,.txt
FINISH
!
*ENDDO

```

Poisson Ratio Sensitivity Solutions

Change in CG Properties

```

!Start 3D HIPped Tensile Analysis
!Requires geometry, elements, and material properties be input previously
/SOLU                           !Start solver
OUTRES,ERASE                   !Erase output files
FINISH                          !Exit solver
*DIM,Fname,CHAR,30,1,1,,,      !Create a vector named Fname
*CREATE,ansuitmp               !Create an input macro file
*VREAD,Fname,'Fname','txt',' ',IJK,30,1,1,, !Read from Fname.txt to Fname vector
(A)                            !Read input in character format
*END                            !Close the macro file
/INPUT,ansuitmp                !Switch the input file for following commands
FINISH                          !Finish the file input

!Apply boundary conditions and input load file
/PREP7                          !Start the preprocessor
!
!Apply symmetry to x, y, and z boundaries at 0
NSSEL,S,LOC,X,0.0              !Select nodes at x = 0.0
DSYM,SYMM,X                    !Apply symmetry condition

```

```

NSEL,S,LOC,Y,0.0          !Select nodes at y = 0.0
DSYM,SYMM,Y              !Apply symmetry condition
NSEL,S,LOC,Z,0.0          !Select nodes at z = 0.0
DSYM,SYMM,Z              !Apply symmetry condition
ALLSEL,ALL               !Reselect all nodes
FINISH                   !Exit preprocessor

!Read input file to 2D array
*DIM,loads,ARRAY,30,2,1
*CREATE,ansuitmp
*VREAD,loads,'loads','txt',,JIK,2,30,1
(F9.7,2X,F10.7)
*END
/INPUT,ansuitmp
FINISH

*DO,ii,1,2,1
*DO,jj,1,30,1
loads(jj,ii,1)=loads(jj,ii,1)*(6.25/3.00)
*ENDDO
*ENDDO
!_____!

!Iterate through the loads from the global model
*DO,q,2,30,1
FINISH                  !Exit solver
/FILNAME,Fname(q,1,1),1 !Create output filename

/PREP7                 !Start the preprocessor
ALLSEL,ALL             !Select all nodes and elements

!Apply displacements from local model
NSEL,S,LOC,X,0.01       !Select nodes at x = 0.01
DDELE,ALL,UX            !Delete existing displacements
D,ALL,UX,loads(q,1,1)   !Displace selected nodes
NSEL,S,LOC,Y,0.01       !Select nodes at y = 0.01
DDELE,ALL,UY            !Delete existing displacements
D,ALL,UY,loads(q,2,1)   !Displace selected nodes
NSEL,S,LOC,Z,0.01       !Select nodes at z = 0.01
DDELE,ALL,UZ            !Delete existing displacements
D,ALL,UZ,loads(q,2,1)   !Displace selected nodes
ALLSEL,ALL              !Reselect all nodes
FINISH                 !Exit the preprocessor

!Solve the load step
/SOLU                   !Start the solver
NSUBST,100              !Specify 100 substeps
SOLVE                   !Solve the load step
FINISH                 !Exit the solver

/POST1                  !Start the Postprocessor
ALLSEL,ALL              !Selects all elements and nodes
ESEL,S,MAT,,2           !Picks all CG elements
ETABLE,cgs1,S,EQV       !Create element table with equivalent stresses (cg mat 2)
FINISH                 !Exit the post-processor

/PREP7                 !Start the pre-processor
ESEL,R,ETAB,cgs1,295.0  !Pick elements that have exceeded 295.0 stress (yield)
EMODIF,ALL,MAT,4        !Change yielding CG elements to material 4
FINISH                 !Exit the pre-processor

/POST1                  !Start the postprocessor
ESEL,S,MAT,,4           !Picks all yielded CG elements
ETABLE,cgs2,S,EQV       !Create element table with equivalent stresses (cg mat 4)
ESEL,R,ETAB,cgs2,390.0  !Pick elements that have exceeded 390.0 stress (UTS)
FINISH                 !Exit the postprocessor

```

```

/SOLU                                !Start the solver
EKILL,ALL                            !Kills selected elements
FINISH                               !Exit the solver

/PREP7                               !Start the preprocessor
EMODIF,ALL,MAT,6                     !Change killed CG elements to material 6
FINISH                               !Exit the preprocessor

/POST1                               !Start the postprocessor
ALLSEL,ALL                           !Selects all elements and nodes
ESEL,S,MAT,,1                        !Picks all UFG elements
ETABLE,ufgs,S,EQV                   !Create element table with the equivalent stresses
ESEL,R,ETAB,ufgs,690.0              !Pick elements that have exceeded 690.0 stress

/SOLU                                !Start the solver
EKILL,ALL                            !Kills selected elements
FINISH                               !Exit the solver

/PREP7                               !Start the preprocessor
EMODIF,ALL,MAT,5                     !Changes killed UFG elements to material 5
FINISH                               !Exit the preprocessor

SAVE,,,ALL                           !Save the current solution to b,c,d,...,txt
FINISH                               !Exit the save
*ENDDO

```

Change in UFG Properties

```

!Start 3D HIPped Tensile Analysis
!Requires geometry, elements, and material properties be input previously

/SOLU                                !Start solver
OUTRES,ERASE                         !Erase output files
FINISH                               !Exit solver
*DIM,Fname,CHAR,30,1,1,,            !Create a vector named Fname
*CREATE,ansuitmp                     !Create an input macro file
*VREAD,Fname,'Fname','txt',' ',,IJK,30,1,1,, !Read from Fname.txt to Fname vector
(A)                                  !Read input in character format
*END                                  !Close the macro file
/INPUT,ansuitmp                      !Switch the input file for following commands
FINISH                               !Finish the file input

!Apply boundary conditions and input load file
/PREP7                               !Start the preprocessor
!_____!
!Apply symmetry to x, y, and z boundaries at 0
NSEL,S,LOC,X,0.0                    !Select nodes at x = 0.0
DSYM,SYMM,X                          !Apply symmetry condition
NSEL,S,LOC,Y,0.0                    !Select nodes at y = 0.0
DSYM,SYMM,Y                          !Apply symmetry condition
NSEL,S,LOC,Z,0.0                    !Select nodes at z = 0.0
DSYM,SYMM,Z                          !Apply symmetry condition
ALLSEL,ALL                          !Reselect all nodes
FINISH                               !Exit preprocessor

!Read input file to 2D array
*DIM,loads,ARRAY,30,2,1
*CREATE,ansuitmp
*VREAD,loads,'loads','txt',,JIK,2,30,1
(F9.7,2X,F10.7)
*END
/INPUT,ansuitmp
FINISH

*DO,ii,1,2,1
*DO,jj,1,30,1

```

```

loads(jj,ii,1)=loads(jj,ii,1)*(6.25/3.00)
*ENDDO
*ENDDO
!_____!

!Iterate through the loads from the global model
*DO,q,1,30,1
FINISH                                !Exit solver
/FILNAME,Fname(q,1,1),1             !Create output filename

/PREP7                                !Start the preprocessor
ALLSEL,ALL                           !Select all nodes and elements

!Apply displacements from local model
NSEL,S,LOC,X,0.01                    !Select nodes at x = 0.01
DDELE,ALL,UX                         !Delete existing displacements
D,ALL,UX,loads(q,1,1)                !Displace selected nodes
NSEL,S,LOC,Y,0.01                    !Select nodes at y = 0.01
DDELE,ALL,UY                         !Delete existing displacements
D,ALL,UY,loads(q,2,1)                !Displace selected nodes
NSEL,S,LOC,Z,0.01                    !Select nodes at z = 0.01
DDELE,ALL,UZ                         !Delete existing displacements
D,ALL,UZ,loads(q,2,1)                !Displace selected nodes
ALLSEL,ALL                           !Reselect all nodes
FINISH                                !Exit the preprocessor

!Solve the load step
/SOLU                                 !Start the solver
NSUBST,100                           !Specify 100 substeps
SOLVE                                 !Solve the load step
FINISH                                !Exit the solver

/POST1                                !Start the Postprocessor
ALLSEL,ALL                           !Selects all elements and nodes
ESEL,S,MAT,,2                        !Picks all CG elements
ETABLE,cgs,S,EQV                     !Create element table with equivalent stresses
ESEL,R,ETAB,cgs,390.0                !Pick elements that have exceeded 390.0 stress (UTS)
FINISH                                !Exit the post-processor

/SOLU                                 !Start the solver
EKILL,ALL                            !Kills selected elements
FINISH                                !Exit the solver

/PREP7                                !Start the preprocessor
EMODIF,ALL,MAT,6                     !Change killed CG elements to material 6
FINISH                                !Exit the preprocessor

/POST1                                !Start the postprocessor
ALLSEL,ALL                           !Selects all elements and nodes
ESEL,S,MAT,,1                        !Picks all UFG elements
ETABLE,ufgs1,S,EQV                  !Create element table with the equiv stresses (ufg mat 1)
FINISH                                !Exit the post-processor

/PREP7                                !Start the pre-processor
ESEL,R,ETAB,ufgs1,580.0              !Pick elements that have exceeded 580.0 stress (yield)
EMODIF,ALL,MAT,3                     !Change yielding UFG elements to material 3
FINISH                                !Exit the pre-processor

/POST1                                !Start the postprocessor
ESEL,S,MAT,,3                        !Picks all yielded UFG elements
ETABLE,ufgs2,S,EQV                  !Create element table with the equiv stresses (ufg mat 3)
ESEL,R,ETAB,ufgs2,690.0              !Pick elements that have exceeded 690.0 stress (UTS)
FINISH                                !Exit the postprocessor

/SOLU                                 !Start the solver
EKILL,ALL                            !Kills selected elements
FINISH                                !Exit the solver

```

```

/PREP7                                !Start the preprocessor
EMODIF,ALL,MAT,5                      !Changes killed UFG elements to material 5
FINISH                                !Exit the preprocessor

SAVE,,,ALL                            !Save the current solution to b,c,d,...,txt
FINISH                                !Exit the save
*ENDDO

```

Change in Bimodal Properties

```

!Start 3D HIPped Tensile Analysis
!Requires geometry, elements, and material properties be input previously
/SOLU                                !Start solver
OUTRES,ERASE                          !Erase output files
FINISH                                !Exit solver
*DIM,Fname,CHAR,30,1,1,,,            !Create a vector named Fname
*CREATE,ansuitmp                      !Create an input macro file
*VREAD,Fname,'Fname','txt',' ',IJK,30,1,1,, !Read from Fname.txt to Fname vector
(A)                                  !Read input in character format
*END                                  !Close the macro file
/INPUT,ansuitmp                      !Switch the input file for following commands
FINISH                                !Finish the file input

!Apply boundary conditions and input load file
/PREP7                                !Start the preprocessor
!
!Apply symmetry to x, y, and z boundaries at 0
NSEL,S,LOC,X,0.0                     !Select nodes at x = 0.0
DSYM,SYMM,X                          !Apply symmetry condition
NSEL,S,LOC,Y,0.0                     !Select nodes at y = 0.0
DSYM,SYMM,Y                          !Apply symmetry condition
NSEL,S,LOC,Z,0.0                     !Select nodes at z = 0.0
DSYM,SYMM,Z                          !Apply symmetry condition
ALLSEL,ALL                           !Reselect all nodes
FINISH                                !Exit preprocessor

!Read input file to 2D array
*DIM,loads,ARRAY,30,2,1
*CREATE,ansuitmp
*VREAD,loads,'loads','txt',,JIK,2,30,1
(F9.7,2X,F10.7)
*END
/INPUT,ansuitmp
FINISH

*DO,ii,1,2,1
*DO,jj,1,30,1
loads(jj,ii,1)=loads(jj,ii,1)*(6.25/3.00)
*ENDDO
*ENDDO
!

!Iterate through the loads from the global model
*DO,q,1,30,1
FINISH                                !Exit solver
/FILNAME,Fname(q,1,1),1              !Create output filename

/PREP7                                !Start the preprocessor
ALLSEL,ALL                           !Select all nodes and elements

!Apply displacements from local model
NSEL,S,LOC,X,0.01                    !Select nodes at x = 0.01
DDELE,ALL,UX                          !Delete existing displacements
D,ALL,UX,loads(q,1,1)                !Displace selected nodes
NSEL,S,LOC,Y,0.01                    !Select nodes at y = 0.01

```

```

DDELE,ALL,UY          !Delete existing displacements
D,ALL,UY,loads(q,2,1) !Displace selected nodes
NSEL,S,LOC,Z,0.01      !Select nodes at z = 0.01
DDELE,ALL,UZ          !Delete existing displacements
D,ALL,UZ,loads(q,2,1) !Displace selected nodes
ALLSEL,ALL             !Reselect all nodes
FINISH                 !Exit the preprocessor

!Solve the load step
/SOLU                  !Start the solver
NSUBST,100             !Specify 100 substeps
SOLVE                  !Solve the load step
FINISH                 !Exit the solver

/POST1                 !Start the Postprocessor
ALLSEL,ALL             !Selects all elements and nodes
ESEL,S,MAT,,2          !Picks all CG elements
ETABLE,cgs1,S,EQV      !Create element table with equivalent stresses (cg mat 2)
FINISH                 !Exit the post-processor

/PREP7                 !Start the pre-processor
ESEL,R,ETAB,cgs1,295.0 !Pick elements that have exceeded 295.0 stress (yield)
EMODIF,ALL,MAT,4       !Change yielding CG elements to material 4
FINISH                 !Exit the pre-processor

/POST1                 !Start the postprocessor
ESEL,S,MAT,,4          !Picks all yielded CG elements
ETABLE,cgs2,S,EQV      !Create element table with equivalent stresses (cg mat 4)
ESEL,R,ETAB,cgs2,390.0 !Pick elements that have exceeded 390.0 stress (UTS)
FINISH                 !Exit the postprocessor

/SOLU                  !Start the solver
EKILL,ALL              !Kills selected elements
FINISH                 !Exit the solver

/PREP7                 !Start the preprocessor
EMODIF,ALL,MAT,6       !Change killed CG elements to material 6
FINISH                 !Exit the preprocessor

/POST1                 !Start the postprocessor
ALLSEL,ALL             !Selects all elements and nodes
ESEL,S,MAT,,1          !Picks all UFG elements
ETABLE,ufgs1,S,EQV     !Create element table with the equiv. stresses (ufg mat 1)
FINISH                 !Exit the post-processor

/PREP7                 !Start the pre-processor
ESEL,R,ETAB,ufgs1,580.0 !Pick elements that have exceeded 580.0 stress (yield)
EMODIF,ALL,MAT,3       !Change yielding UFG elements to material 3
FINISH                 !Exit the pre-processor

/POST1                 !Start the postprocessor
ESEL,S,MAT,,3          !Picks all yielded UFG elements
ETABLE,ufgs2,S,EQV     !Create element table with the equiv. stresses (ufg mat 3)
ESEL,R,ETAB,ufgs2,690.0 !Pick elements that have exceeded 690.0 stress (UTS)
FINISH                 !Exit the postprocessor

/SOLU                  !Start the solver
EKILL,ALL              !Kills selected elements
FINISH                 !Exit the solver

/PREP7                 !Start the preprocessor
EMODIF,ALL,MAT,5       !Changes killed UFG elements to material 5
FINISH                 !Exit the preprocessor

SAVE,,,ALL             !Save the current solution to b,c,d,...,txt
FINISH                 !Exit the save
*ENDDO

```